NASA Conference Publication 10063

Aeropropulsion '91

AEROPROPULSION
1991

Proceedings of a conference held at
NASA Lewis Research Center
Cleveland, Ohio
March 20-21, 1991

NASA

(NASA-CP-10063) AEROPROPULSION 1991 (NASA) 574 p CSCL 21E
N91-20086
--THRU--
N91-20121
Unclas
G3/07 0002046

NASA Conference Publication 10063

Aeropropulsion '91

Proceedings of a conference held at
NASA Lewis Research Center
Cleveland, Ohio
March 20–21, 1991

NASA
National Aeronautics and
Space Administration

Office of Management

Scientific and Technical
Information Division

1991

FOREWORD

In this 50th year since ground was broken for the NACA Aircraft Engine Research Laboratory, Aeropropulsion '91 provides a review of the recent accomplishments of the ongoing program at Lewis Research Center which has provided a foundation for the dominance of U.S. industry in aeropropulsion. The Lewis Aeropropulsion Program has the charter to develop advanced airbreathing propulsion technology for applications of national interest. To implement this program, Lewis concentrates its basic research in the critical disciplines of internal fluid mechanics, instrumentation and control, and structures and materials. These efforts are focused on developing new science, often implemented through advanced computer codes. Carefully designed experiments, often using new nonintrusive measurement techniques, provide the data required to validate the advanced codes. Recent results and planned accomplishments in these areas are reviewed.

From this firm foundation, NASA Aeropropulsion Systems programs investigate technology for a number of applications including propulsion for subsonic and supersonic transports, high performance and hypersonic aircraft, rotorcraft, and general aviation. These programs are accomplished in cooperation with industry and are focused on validating advanced technology for the future engines of U.S. industry. These programs are reviewed with emphasis on accomplishments and future goals.

Aeropropulsion '91, held at Lewis Research Center on March 20 and 21, 1991, reviewed the plans and results for the program described above. Representatives from industry, universities, and other Government agencies attended the conference. This publication contains figures and supporting text for the papers presented at the conference. Authorship credit is listed with each paper in these proceedings.

PRECEDING PAGE BLANK NOT FILMED

CONTENTS

Page

LEWIS AEROPROPULSION TECHNOLOGY: REMEMBERING THE PAST AND CHALLENGING THE FUTURE
Neal T. Saunders and Arthur J. Glassman, NASA LeRC 1-1

OVERVIEW OF SUPERSONIC CRUISE PROPULSION RESEARCH
R.J. Shaw, NASA LeRC 2-1

OVERVIEW OF HIGH PERFORMANCE AIRCRAFT PROPULSION
Peter G. Batterton and Thomas J. Biesiadny, NASA LeRC 3-1

FLOW VISUALIZATION AND HOT GAS INGESTION CHARACTERISTICS OF A VECTORED THRUST STOVL CONCEPT
Albert L. Johns, George H. Neiner, Timothy J. Bencic and David M. Fricker, NASA LeRC; and Jerome T. Kegelman, McDonnell Douglas Research Laboratories 4-1

HIGH ALPHA INLETS
Richard R. Burley and Bernhard H. Anderson, NASA LeRC; and C. Frederic Smith and Gary J. Harloff, Sverdrup Technology, Inc. 5-1

OVERVIEW OF HYPERSONIC/TRANSATMOSPHERIC VEHICLE PROPULSION TECHNOLOGY
John E. Rohde, NASA LeRC 6-1

OVERVIEW OF NASP NOZZLE RESEARCH
Charles J. Trefny, Rickey J. Shyne, and Jeanne D. Carboni, NASA LeRC 7-1

A COMPARISON OF CFD PREDICTIONS AND EXPERIMENTAL RESULTS FOR A MACH 5 INLET
Lois J. Weir, NASA LeRC; and D.R. Reddy, Sverdrup Technology, Inc. 8-1

INTERNAL FLUID MECHANICS RESEARCH
Lonnie Reid and Louis A. Povinelli, NASA LeRC 9-1

INLETS, DUCTS, AND NOZZLES
James R. Scott and John M. Abbott, NASA LeRC 10-1

TURBOMACHINERY
Anthony J. Strazisar, Raymond E. Gaugler, John J. Adamczyk, and Dale J. Arpasi, NASA LeRC; and Isaac Lopez, Propulsion Directorate, U.S. Army Aviation Systems Command 11-1

CHEMICAL REACTING FLOWS
Erwin A. Lezberg, NASA LeRC; Edward J. Mularz, Propulsion Directorate, U.S. Army Aviation Systems Command; and Meng-Sing Liou, NASA LeRC 12-1

PRECEDING PAGE BLANK NOT FILMED

INSTRUMENTATION AND CONTROLS OVERVIEW
Norman C. Wenger, NASA LeRC 13-1

PROPULSION INSTRUMENTATION RESEARCH
William C. Nieberding, Daniel J. Lesco, and W. Dan Williams, NASA LeRC 14-1

HIGH TEMPERATURE ELECTRONICS
Gary T. Seng, NASA LeRC 15-1

FIBER-OPTIC-BASED CONTROLS
Gary T. Seng, NASA LeRC 16-1

ADVANCED AEROPROPULSION CONTROLS TECHNOLOGY
Carl F. Lorenzo, NASA LeRC 17-1

OVERVIEW OF STRUCTURES RESEARCH
Larry D. Pinson, NASA LeRC 18-1

PROPULSION AEROELASTICITY, VIBRATION CONTROL, AND DYNAMIC SYSTEM MODELING
Bruce M. Steinetz, NASA LeRC 19-1

COMPUTATIONAL SIMULATION OF PROPULSION STRUCTURES PERFORMANCE AND RELIABILITY
Dale A. Hopkins, NASA LeRC 20-1

TRANSITION-TO-PRACTICE TECHNOLOGIES FOR BRITTLE MATERIALS
Edward R. Generazio, NASA LeRC 21-1

PROGRESS IN MODELING DEFORMATION AND DAMAGE
Alan D. Freed and Steven M. Arnold, NASA LeRC 22-1

OVERVIEW OF LEWIS MATERIALS RESEARCH - CONTRIBUTIONS, CURRENT EFFORTS, AND FUTURE DIRECTIONS
Salvatore J. Grisaffe and Carl E. Lowell, NASA LeRC 23-1

ADVANCED HIGH TEMPERATURE ENGINE MATERIALS TECHNOLOGY PROGRAM
Hugh R. Gray, NASA LeRC 24-1

COMPUTATIONAL MATERIALS SCIENCE
AN EXAMPLE: NUMERICAL MODELING OF CHEMICAL VAPOR DEPOSITION PROCESSING OF ADVANCED FIBERS
Arnon Chait, Suleyman Gokoglu, and Maria Kuczmarski, NASA LeRC; Pang Tsui, Sverdrup Technology, Inc.; Lisa Veitch, NASA LeRC; and David Thompson and Ronald Gaug, The University of Akron 25-1

OVERVIEW OF ROTORCRAFT AND GENERAL AVIATION PROPULSION TECHNOLOGY
John J. Coy and Joseph D. Eisenberg, NASA LeRC 26-1

TURBOMACHINERY AND COMBUSTOR TECHNOLOGY FOR SMALL ENGINES
Lawrence J. Bober and Richard W. Niedzwiecki, NASA LeRC 27-1

ROTARY ENGINE TECHNOLOGY
Edward A. Willis and John J. McFadden, NASA LeRC 28-1

ADVANCED ROTORCRAFT TRANSMISSION TECHNOLOGY
David G. Lewicki, Propulsion Directorate, U.S. Army Aviation Systems Command 29-1

OVERVIEW OF SUBSONIC TRANSPORT PROPULSION TECHNOLOGY
John F. Groeneweg, NASA LeRC 30-1

ULTRA-HIGH BYPASS RESEARCH
Christopher J. Miller, James H. Dittmar, and Robert J. Jeracki, NASA LeRC 31-1

HIGH-EFFICIENCY CORE TECHNOLOGY
Robert M. Plencner and Gerald Knip, Jr., NASA LeRC 32-1

MULTIDISCIPLINARY RESEARCH OVERVIEW (IMPTET/NPSS)
Lester D. Nichols and Susan M. Johnson, NASA LeRC 33-1

NASA'S AIRCRAFT ICING TECHNOLOGY PROGRAM
John J. Reinmann, NASA LeRC 34-1

RECENT ADVANCES IN LEWIS AEROPROPULSION FACILITIES
Frank J. Kutina, Jr., NASA LeRC 35-1

N91-20087

LEWIS AEROPROPULSION TECHNOLOGY: REMEMBERING THE PAST AND CHALLENGING THE FUTURE

Neal T. Saunders and Arthur J. Glassman
NASA Lewis Research Center
Cleveland, Ohio

It was on January 23, 1941, less than two years after the first flight of a jet-propelled aircraft, that George Lewis, Director of Aeronautical Research for NACA, broke ground in Cleveland for the NACA Aircraft Engine Research Laboratory (AERL), which was to bear his name after his death in 1948. Originally envisioned as a laboratory for fundamental research on piston engines, the new NACA laboratory never actually fulfilled this role. From the first test in 1942 until the war's end in 1945, primary emphasis was on trouble-shooting to solve the problems of engines in production for the war effort. By the end of the war, the transition from piston to jet propulsion was well underway, and with it went the direction of the laboratory's program.

In this presentation, we will revisit some of our major accomplishments over the past fifty years and pick up the gauntlet to meet the challenges of the future. From piston engines through environmentally acceptable high-speed propulsion systems, efforts have included and will continue to include discipline, component, and engine activities along with provision of unique facilities to carry out our programs.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Remembering the Past

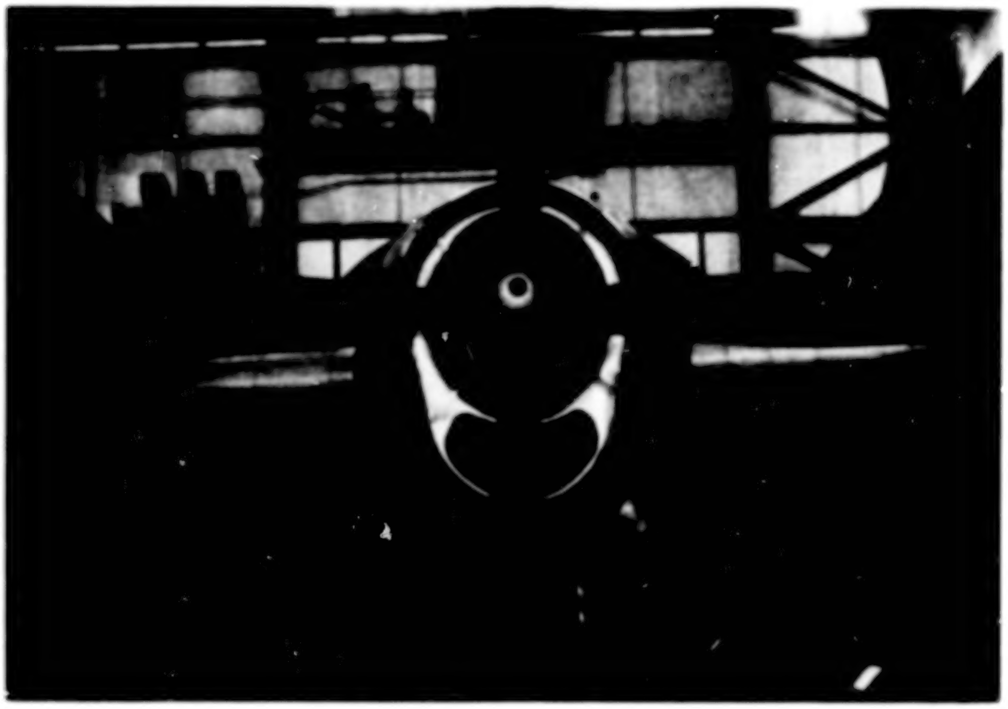
- Historical facilities
- Component technologies
- Environmental acceptability
- Energy efficiency

CD-91-[illegible]

After addressing key problems (turbocharging, carburetion, cooling) plaguing the advanced piston engines being produced for the war, research focus shifted to gas turbine engines for turbojet, turbofan, and turboprop propulsion systems. Advances were made in the understanding and state of the critical discipline technologies (materials, internal fluid mechanics, structures, instrumentation, controls), the capability and efficiency of components (propellers, compressors, turbines, combustors, inlets, nozzles), and the fuel efficiency and environmental acceptability (noise, emissions) of engines. Let us now recall some of the more notable aeropropulsion technology achievements of the Lewis Research Center over the past half century.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Piston Engines and the Altitude Wind Tunnel



CD-91-[illegible]

After much debate, the full-scale Altitude Wind Tunnel (AWT) was included as one of the first facilities built at AERL. The concern was that the AWT would be used for engine development, which should be the domain of the engine companies. Fortunately, the tunnel was built, as it was given credit by some for shortening the war.

Although the AWT was designed for piston engine testing, the first tests were conducted in February 1944 on the secret I-16 turbojet engine developed by General Electric from the Whittle prototype. The first official tests in AWT were with the Wright R-3350 piston engine urgently needed for the B-29 bomber. This engine, which had been rushed into production, had many problems, the foremost of which was random poor fuel distribution causing unpredictable cylinder overheating. This was corrected by use of fuel injection instead of a carburetor. This and other fixes developed in the Lewis AWT contributed to the B-29's, including the Enola Gay's, being able to fly above the enemy's altitude capability while performing its missions.

Icing Research Tunnel

INTERNATIONAL HISTORIC
MECHANICAL ENGINEERING LANDMARK
ICING RESEARCH TUNNEL
LEWIS RESEARCH CENTER OF NASA
CLEVELAND OHIO
1944

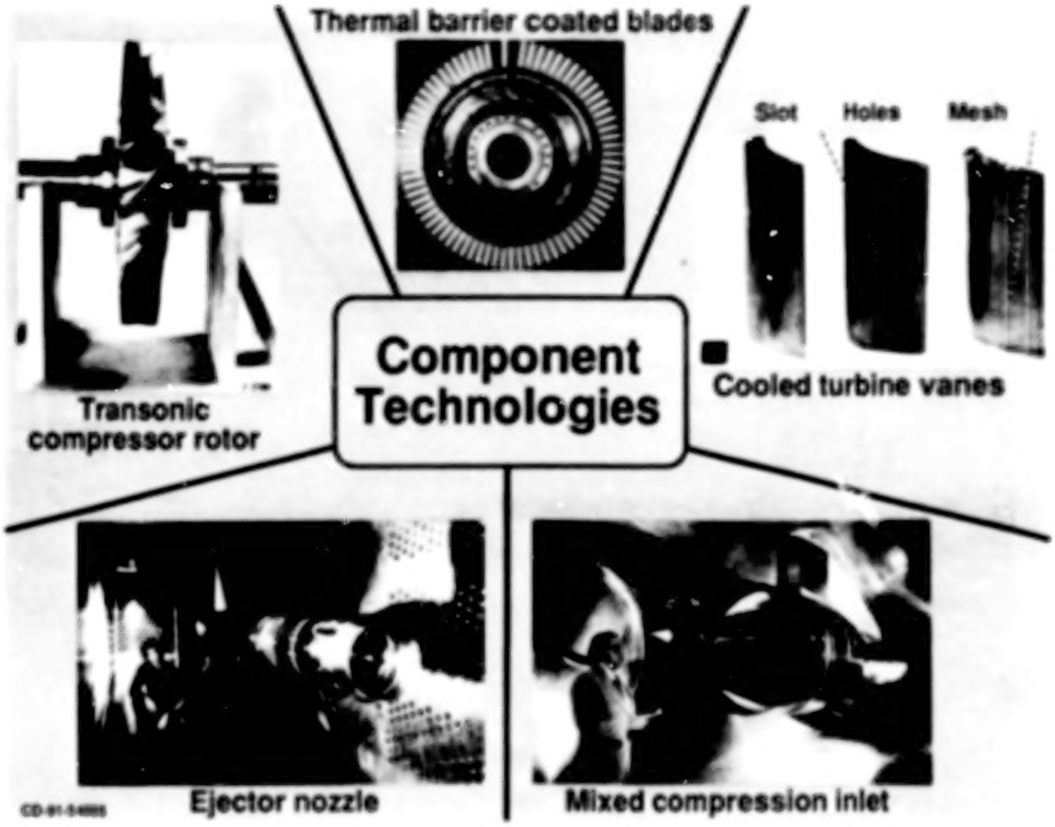
CD-91-54004

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Responding to a critical need expressed by the Army Air Force and aircraft manufacturers, the NACA initiated research on aircraft icing and directed that an Icing Research Tunnel (IRT) be added to the AWT under construction. Developing this unique capability required unusual ingenuity to solve critical technical problems. The first icing test was run on June 9, 1944. Today, the IRT remains the world's largest refrigerated icing tunnel.

The heat exchanger design was revolutionary; it can maintain a spatially uniform air temperature over a very large range of airspeed and temperature, even when subjected to severe icing. Another major innovation was the spray system. In 1943, no one knew just how small natural cloud droplets were or how to measure them, let alone how to duplicate them. By the mid-1940's, the nature of the droplets had been defined and by early 1950, the spray system was capable of producing droplets small enough to reproduce realistic icing patterns.

Between 1950 and 1958, the IRT was used for extensive testing of aircraft components. The anti-icing technology used on today's commercial transports was largely developed in the IRT. Over the next 20 years, with icing problems seemingly solved, the tunnel was used very little. In 1978, NASA re-instituted an icing research program to address the needs for new aircraft designs. After a major renovation in 1986, the IRT is again in heavy demand from government and industry. In 1987, the American Society of Mechanical Engineers (ASME) designated the Lewis Research Center's Icing Research Tunnel an International Historic Mechanical Engineering Landmark for its leading role in making aviation safer for everyone.



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Much research has been devoted to understanding, extending, and documenting the capabilities and operating limits of engine components and associated disciplines. Operational increases in flight speed and in engine temperature and pressure ratio came about only after the components were able to efficiently accommodate them. Early axial-flow compressors were limited to stage pressure ratios of 1.1 to 1.2 because of the design limitation to subsonic flow. Blade shapes capable of transonic operation were developed to produce a stage pressure ratio approaching 1.4, thus resulting in substantial stage reduction. Turbine inlet temperature capability has increased substantially as a result of high-temperature turbine materials and turbine cooling. Research in the areas of alloy development, failure mechanisms, and protective coatings, including thermal barriers, has contributed to numerous heat-resistant, long-life turbine disk and blade materials. Advances in casting technology enabled the development of advanced cooling concepts. A systematic program of cascade and turbine tests documented the effects of number, location, size, and angle of cooling holes, thus providing an extensive data base for the design of aerodynamically efficient cooled blading.

Extensive studies of characteristics such as efficiency, stability, and distortion for a wide variety of supersonic inlets have produced an inlet design and control data base that has been used for the design of inlets for many military aircraft. Many of these inlets were developed using the Lewis tunnels. Exhaust nozzle testing in both isolated and installed configurations has been aimed primarily at noise reduction and at variable geometry for good performance at both subsonic and supersonic flight. This data has also been used extensively by industry.

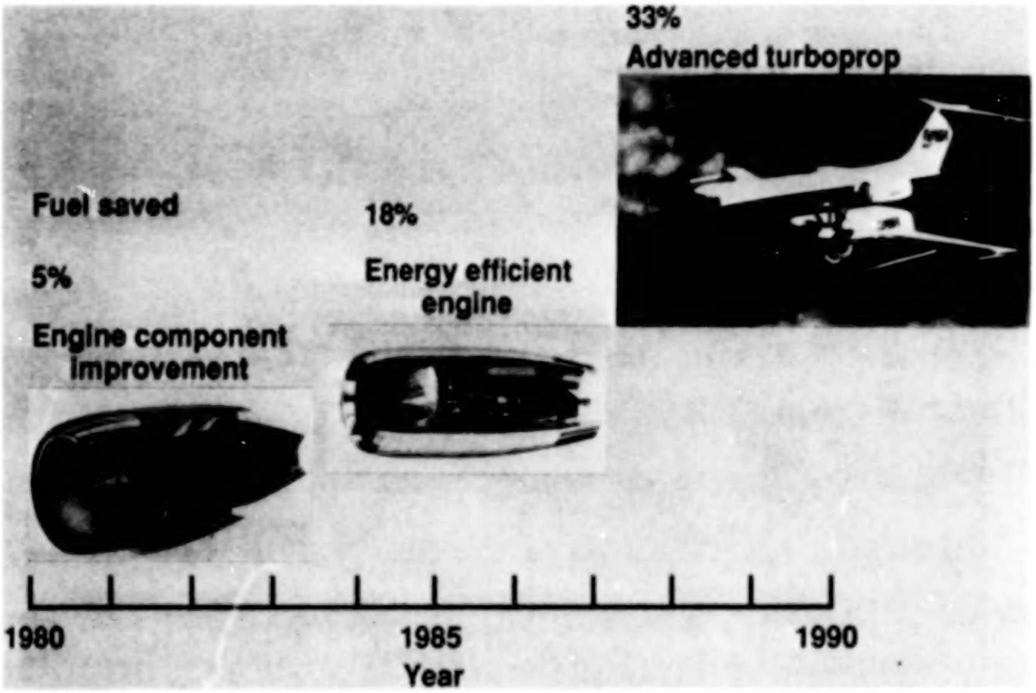
DC-9 refan airplane
Environmental Acceptability
Quiet engine
Staged combustors
Swirl–can combustor

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The phenomenal growth of air traffic in the early 1960's intensified an already growing concern about aircraft noise and emissions and led to programs addressing these environmental issues. The Quiet Engine was an experimental engine whose primary objective was to provide a full-size demonstration of the effectiveness of noise reduction techniques. General Electric designed and built the engine, while Boeing provided the nacelle. Engine tests at NASA Lewis demonstrated that the Quiet Engine technologies, compared to the current 707 and DC-8 aircraft engines, could provide about 20 EPNdB noise reduction without fan acoustic treatment and nearly 30 EPNdB with an acoustic nacelle. The Refan Program was a demonstration of using noise abatement technologies to quiet the narrow-body fleet by retrofit. On the basis of Pratt & Whitney and NASA engine tests (JT8D), Boeing ground tests (727 nacelles), and Douglas flight tests (DC-9), it was shown that 5 to 10 EPNdB takeoff and landing noise reduction and greater than 60 percent reduction in area exposed to 90 dB and higher noise levels were achievable.

Lewis has played a major role in the evolution of low-pollutant combustor technology. The swirl-can combustor concept, which operates leaner than a conventional combustor, demonstrated a twofold reduction in nitrogen oxide emissions. The Experimental Clean Combustor Program, in which Pratt & Whitney and General Electric participated, developed the technology for staged combustion concepts that provided a twofold reduction in nitrogen oxides at takeoff and about a tenfold reduction in carbon monoxide and hydrocarbon emissions at idle. A staged combustor will be used in the next generation of civil high-bypass engines, which is scheduled to enter service in the next few years.

Energy Efficient Propulsion Technology



CD-91-54007

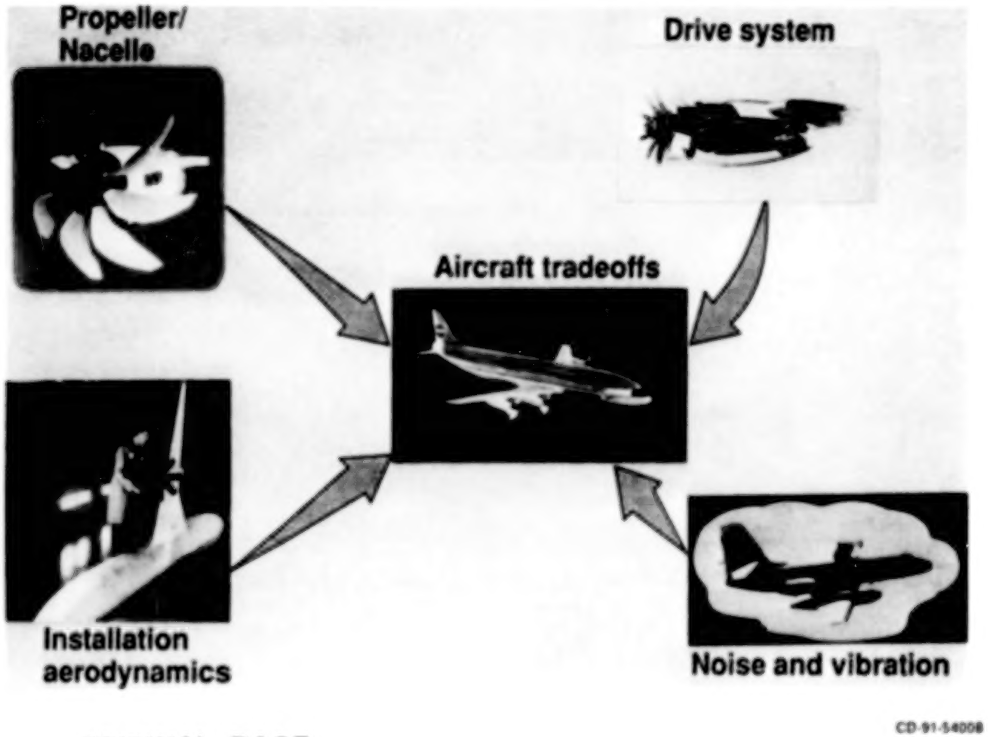
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

In January 1975, the U.S. Senate requested NASA to establish an advanced technology program for aircraft fuel conservation. The NASA Aircraft Energy Efficiency Program was thus initiated, with three of its six elements focusing on propulsion: two addressing turbofan engines and one addressing turboprops. The turbofan elements were Engine Component Improvement (ECI) and Energy Efficient Engine (EEE). General Electric and Pratt & Whitney participated in both of these programs.

The ECI project was expected to provide near term fuel savings of 5 percent for the existing commercial fleet. The performance improvement was to come from improved component aerodynamics, reduced leakages, and reduced cooling. Concepts implemented on the basis of fuel savings potential and favorable benefit/cost ratio resulted in total fuel savings meeting the 5 percent goal. NASA was reimbursed for program costs based on commercial sales of the upgraded engines incorporating ECI technologies.

The objective of the EEE project was to provide a technology base for a new generation of fuel efficient turbofan engines with specific fuel consumption at least 12 percent below that of then-current high-bypass engines. This program was based on "clean-sheet" design engines. Advanced technologies were evaluated in sub-scale and full-scale rigs and then incorporated into both core and integrated core/low spool engines for final evaluation. Fuel savings were achieved through use of an advanced cycle, improved components, and a mixer. If all the EEE technologies were applied to an optimized new engine, the fuel savings benefit would be about 18 percent. Substantial portions of the fuel savings of current technology engines can be attributed to EEE technology developments.

Advanced Turboprop Enabling Technology



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The third propulsion element of the Aircraft Energy Efficiency Program was the Advanced Turboprop, which provided the highest potential fuel savings - about 30 to 35 percent relative to then-current high-bypass turbofan engines. To address this most challenging concept, the approach followed a logic path that started with analyses and proceeded to design codes and designs based on scale-model tests. Large-scale systems were then designed, ground tested, and ultimately flight tested. All key system elements were considered, and geared single-rotation as well as direct-drive and geared counterrotation concepts were included.

Achievement of high propeller efficiency and low noise combined with structural integrity were key to program success. These requirements were addressed and demonstrated early in the program through a combination of scale-model tests and code development. Installing the turboprop on a wing could reduce the efficiencies of both the wing and the engine, thus severely limiting any fuel efficiency gains. Model tests were used to study installation effects in Ames and Langley wind tunnels. Because of power levels and number of blades, neither the drive system's gearbox nor its blade pitch-change mechanism could be of conventional design. Lewis along with several major engine companies had to develop these technologies. There were major concerns about effects of turboprop noise and vibration on both interior and community environments. Under Lewis direction, Dryden took responsibility for model-source-noise flight testing and Langley for aircraft interior environment analysis and noise attention. With the enabling technology developed, large-scale system design and testing could be undertaken.

Advanced Turboprop Proof of Concept

ADVANCED TURBOPROP PROJECT
NASA LEWIS RESEARCH CENTER

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Beginning in August 1986, the advanced turboprop propulsion concept was proven by flight programs using large-scale hardware. The NASA/General Electric/Boeing flight test and the General Electric/McDonnell Douglas flight test used the Unducted Fan as a proof-of-concept demonstrator for the gearless counterrotating concept and provided in-flight performance and acoustic data. The NASA/Lockheed-Georgia Propfan Test Assessment flight test verified the structural integrity and acoustic characteristics of the single-rotation Large-Scale Advanced Propfan built by Hamilton Standard. Encouraged by these results, Pratt & Whitney/Allison built a geared counterrotating engine, based on design data acquired in Lewis-funded Allison gearbox and Hamilton Standard/United Technologies Research Center model tests, and flight tested it on a McDonnell Douglas aircraft.

In recognition of the success of this effort, the NASA Lewis Research Center and the NASA/industry advanced turboprop team were awarded the 1987 Collier Trophy, which is awarded yearly for the greatest achievement in aeronautics or astronautics. Studies, model tests, and flight tests have shown that turboprops with thin, swept, highly loaded blades can operate at high speeds (Mach 0.65 to 0.85) and reduce fuel consumption 25 to 30 percent relative to advanced turbofans and 40 to 50 percent relative to today's aircraft.

Challenging the Future

- Vision/thrusts
- Focused programs
- Basic- and multi-discipline technologies
- Facilities

CD-91-54010

In the future, we at NASA Lewis Research Center will continue to develop and assist the application of advanced technologies required for high performance propulsion systems for civilian and military aircraft. We will accomplish this by conducting focused research and technology efforts related to the prime propulsion needs of subsonic, supersonic cruise, rotorcraft/general aviation, high-performance military, and hypersonic/TAV aircraft. In addition, we will give continuing emphasis to the critical basic disciplines of internal fluid mechanics, instrumentation and controls, materials, and structures, and to efforts aimed at the complex interactions among multiple disciplines. Further, we must assure that our facilities meet our research needs.

Our Vision

To be the Nation's pathfinder in innovative aerospace propulsion R&T

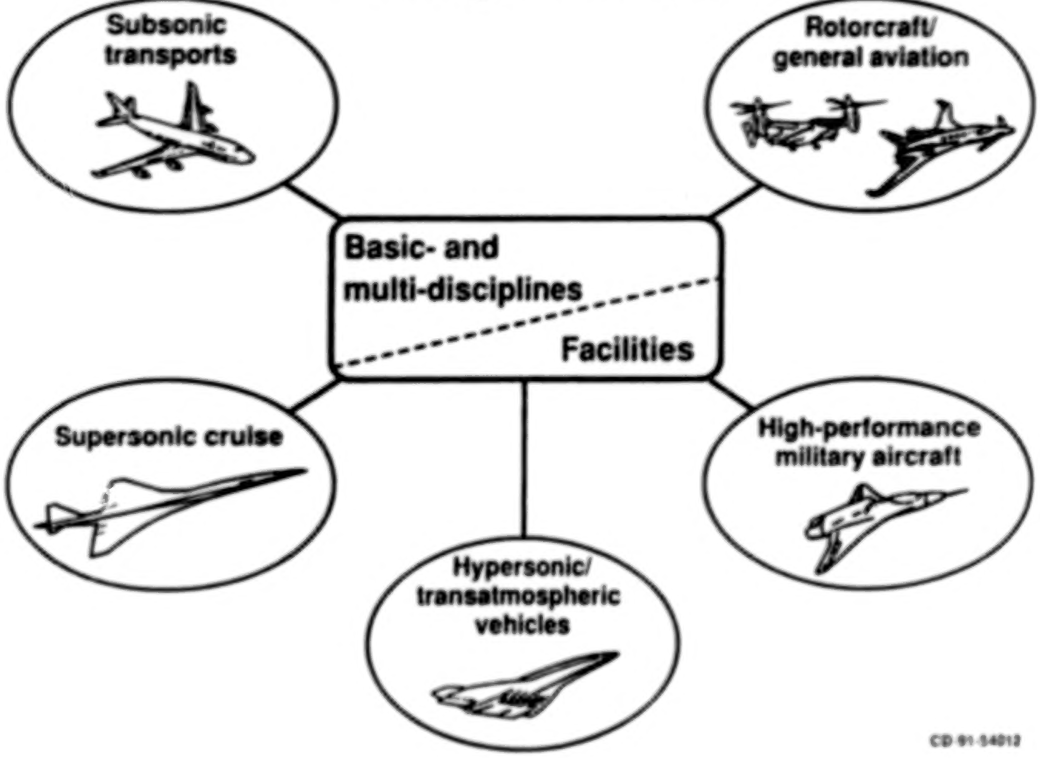
Commits us to:

- **Technology advances**
- **World-class computational and experimental facilities**
- **A challenging working environment**
- **Strong collaborative relationships with external partners**

CD-91-54011

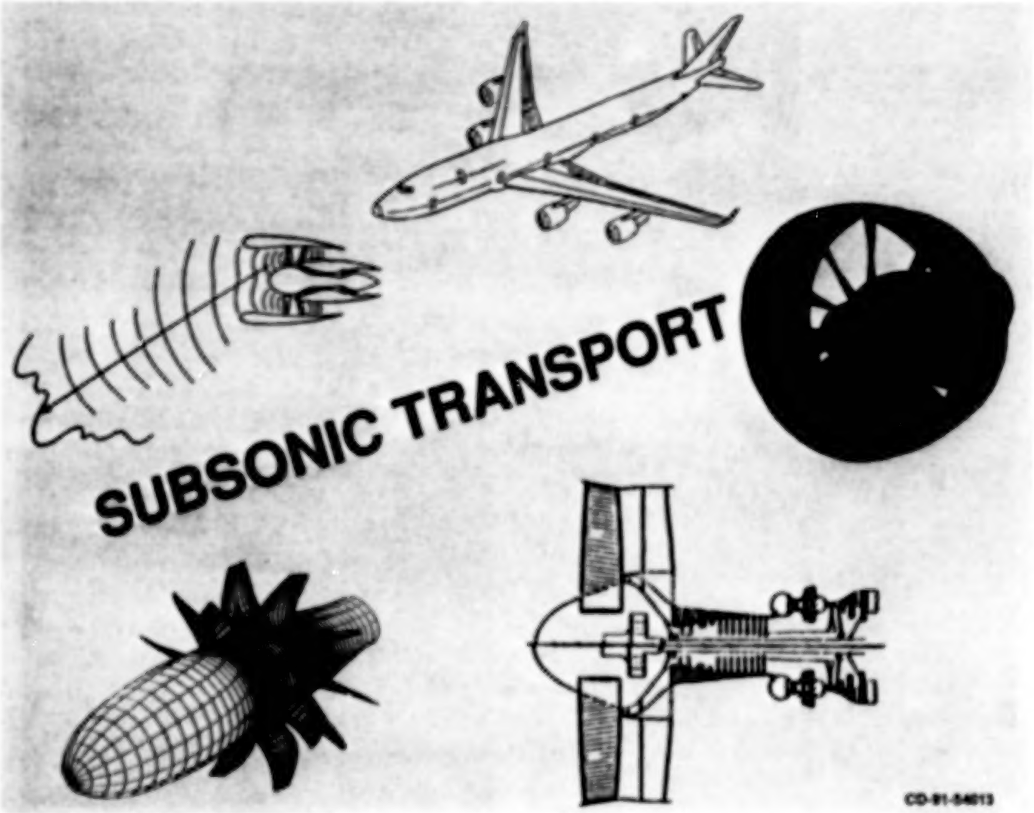
The Lewis Research Center aeropropulsion vision, presented above, commits us to (1) conduct high-payoff innovative research to provide leading-edge propulsion technology advances, (2) develop, maintain, and effectively utilize world-class computational and experimental facilities, (3) provide an exciting action-oriented working environment which attracts, develops, and retains outstanding scientists and engineers, (4) promote strong collaborative relationships with industry, universities, and other government agencies, and (5) fulfill our programmatic commitments at the same time.

Strategic Thrusts


Subsonic transports
Rotorcraft/ general aviation
Basic- and multi-disciplines
Facilities
Supersonic cruise
High-performance military aircraft
Hypersonic/ transatmospheric vehicles

CD-91-54012

The NASA aeronautics strategic thrusts provide the focus for the specific elements of the aeropropulsion program. We implement our propulsion technology efforts through five vehicle-focused elements (subsonic transports, supersonic cruise, hypersonic/ transatmospheric vehicles, high-performance military aircraft, and small-engine technologies for rotorcraft/general aviation aircraft) plus some generic technology elements involving both basic-disciplinary and multi-disciplinary research. About two-thirds of our resources are directed at the five vehicle-focused elements, and the remaining one-third is invested in our generic research and our aeropropulsion facilities that are essential to all of our efforts.



The goal of this program is to achieve revolutionary advances in efficiency and environmental acceptability of subsonic transport aircraft. There is some remaining work aimed at maturing turboprop technology and transferring it to U.S. industry. Advanced concepts such as forward sweep for counterrotation are being explored.

Major elements of the subsonic transport program involve ducted prop/fan technology and high thermal efficiency cores. Emphasis is on noise reduction, with a goal of 6 to 7 EPNdB below FAR 36 Stage 3, to be achieved by fan source noise reduction and nacelle acoustic treatment. Through component/wind tunnel tests and aerodynamic/acoustic analytical modeling, a design methodology will be developed and then demonstrated in large scale. High thermal efficiency for an ultra-high-bypass engine core is achieved by increasing overall pressure ratio and turbine temperature in concert with improving turbomachine efficiencies and reducing/eliminating turbine cooling. In addition, novel concepts involving the use of offset cores, intercooling, and reheating are being studied for potential benefits.

SUPERSONIC CRU
F/A RATIO

CD-90-45899

The goal of this program is to provide enabling propulsion technology for environmentally compatible and economically viable supersonic transports. In the near term, efforts are directed toward developing the technology base required for viable solutions to the environmental and economic barrier issues related to a proposed High-Speed Civil Transport (HSCT). Specifically, environmental programs are aimed at low emissions combustor technology that will result in no measurable impact on the ozone layer and low noise nozzle technology that will contribute to complying with FAR 36 Stage 3 noise rules. To address the economic issue, programs will develop and demonstrate the enabling materials and critical component technologies.

Far-term efforts will be directed at the development of advanced technologies for enhancing the performance of supersonic cruise propulsion systems. One example of such an effort is the development of supersonic throughflow fan technology to provide a basis for alternate propulsion system designs.

CRAFT/GA
CD-90-46802

The goal of this focus is to provide innovative technologies to improve rotorcraft/ general aviation propulsion systems in order to strengthen the nation's competitive stance in the world market. A reduction of 30 percent in the fuel consumption of small engines is achievable through use of advanced cycles having higher temperature and pressure, improved turbomachinery components, and ceramic materials. Contributions to fuel savings, safety, and reliability can be made with advanced technology transmissions. Advanced anti-icing/deicing technology will contribute to aircraft safety.

In the area of small turbine engines, the research will address centrifugal compressors, combustors, and uncooled/cooled radial turbines. Detailed measurements will provide both an understanding of the flow physics and verification of predictive CFD capabilities. Combustion technology efforts will address very low pattern factors. Use of ceramic components to 2500 °F will be demonstrated. A rotary engine having low fuel consumption at high altitude is being explored. Advanced rotorcraft transmissions achieving 25 percent weight reduction, 10 dB noise reduction, and 5000 hr between removals will be demonstrated.

HIGH PERFORMANCE AIRCRAFT

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The goal of this program is to provide propulsion technology options for revolutionary performance capabilities in future high-performance military aircraft. Technologies and test facilities for future prototype vehicle programs will be developed. To minimize risk and assure success in flight demonstration programs, appropriate studies and ground-based tests will be performed to validate concepts prior to flight commitment. Participation in joint efforts with other NASA centers, the Department of Defense, and foreign governments are often key elements of these programs.

Primary emphasis is on propulsion systems for short takeoff/vertical landing (STOVL) and high maneuverability aircraft. Specifically, analytical assessment and experimental research will be conducted for unique components and problems such as internal valves and transport ducting, vertical thrusting and vectoring nozzles, hot gas ingestion, controls, and special thrust augmenters.

HYPERSONIC/TAV

CD 90 [illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This focus has as its goal to provide enabling propulsion technology for future development of hypersonic/TAV flight vehicles. Accomplishment will be by developing specific hypersonic propulsion technologies, predictive codes, and an understanding of the governing physics that are needed to support the National Aerospace Plane (NASP) and other hypersonic vehicles.

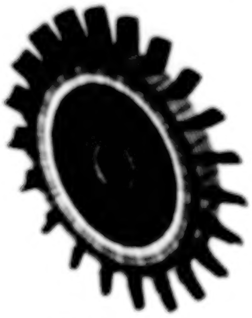
Specific technologies to be addressed include hydrogen cooled engine components, hydrogen production, storage, and handling, inlet performance and operating characteristics, ramjet engine controls, and exhaust nozzle behavior including base burning. Verification of inlet, combustor, and nozzle CFD computer codes will give credence to predictive capabilities in the hypersonic regime.

NASA Lewis is participating with the U.S. Air Force in the High Mach Turbine Engine (HiMaTE) program addressing propulsion at Mach 4 to 6. Studies, component demonstrations, and possible engine tests comprise this program.

Basic Discipline Technologies

Internal fluid mechanics

Materials


Structures

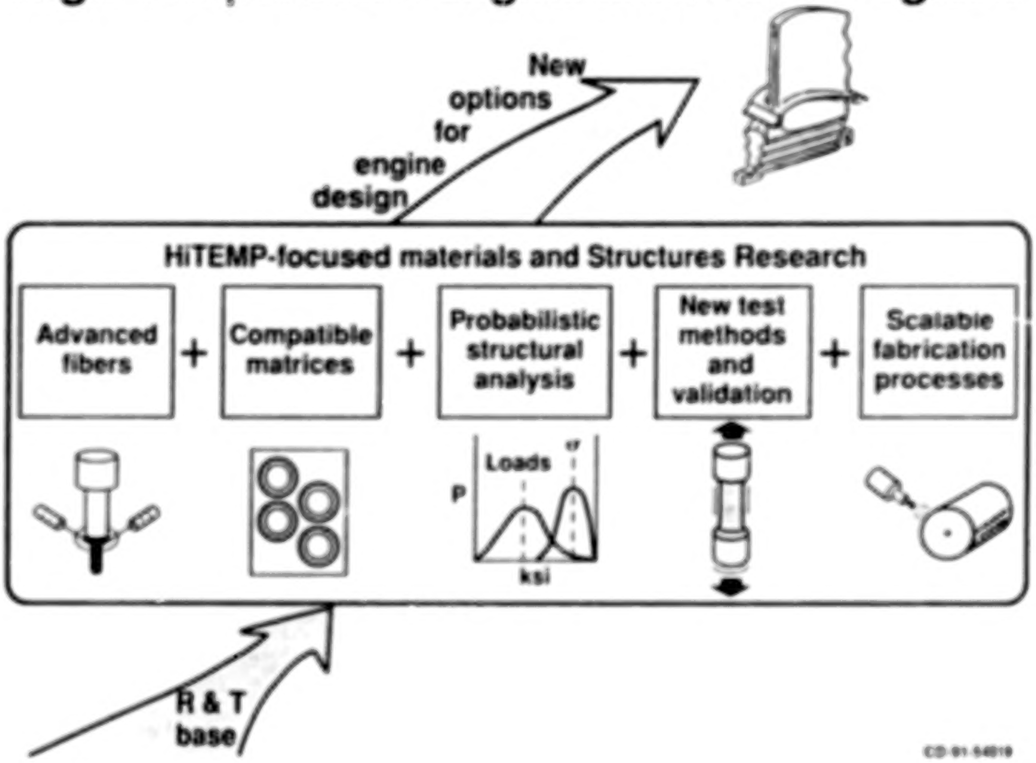
Instrumentation and controls

CD-91-54018

A key part of our strategy for the future is our commitment to build our in-house capabilities in the critical disciplines. This involves strengthening the basic discipline areas of materials, structures, internal fluid mechanics, and instrumentation and controls. Multidisciplinary programs, such as HITEMP and NPSS, are then used to help focus and integrate various disciplinary efforts on key aeropropulsion technology needs.

Advances in the basic disciplines are essential to achieving future advances in aeropropulsion technology. Internal fluid mechanics involves understanding and predicting the mass, momentum, and heat transport within the engine, thus enabling optimization of the aerodynamic design process. Engine materials have paced the progress of propulsion and will continue to do so for the foreseeable future - primarily as a result of increasing temperature capability. Sustained vibrations and large deflections in the engine can lead to loss of performance or of overall structural integrity, thus necessitating the understanding and prediction of structural behavior. Instrumentation technology is addressing both research and operational needs, which involve making measurements of increasing detail in environments of ever increasing severity. Key activities in the controls area involve the use of real-time intelligence to improve propulsion system performance and operability by closed-loop control.

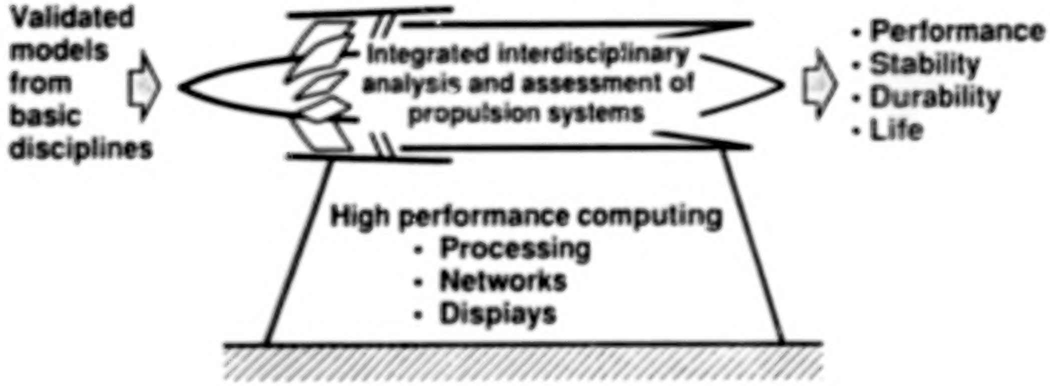
High Temperature Engine Materials Program


New options for engine design
HiTEMP-focused materials and Structures Research
Advanced fibers
+
Compatible matrices
+
Probabilistic structural analysis
+
New test methods and validation
+
Scalable fabrication processes
Loads
σ
P
ksi
R & T base
CD-91-54019

To achieve revolutionary advances in propulsion systems for 21st century transports, high temperature materials have been identified as the key technology to be addressed. NASA's High Temperature Engine Materials Technology Program (HITEMP) is directed towards generating this technology. The HITEMP Program is focusing primarily on lightweight composite materials to gain revolutionary advances in the operating temperature of advanced engines compared to the current state-of-the-art. Emphasis is being placed on polymer matrix composites (PMC's) for potential use in fans, casings, and engine control systems. Intermetallic/metal matrix composites (IMC's/MMC's) are under investigation for application in such areas as the compressor and turbine disks, blades, and vanes, and in the exhaust nozzle. For extremely high temperature applications, ceramic matrix composites (CMC's) are being explored for applications such as liners for the combustor and exhaust nozzle, turbine vanes, and ultimately the turbine blades and disks.

NASA considers this program to be a focused research effort that builds upon our basic research programs and that will feed results into application oriented projects such as the proposed NASA new initiative to develop the technology for a 21st century High Speed Civil Transport (HSCT). The Enabling Propulsion Materials (EPM) program is a major effort in the HSCT program and will utilize materials and structures concepts developed in HITEMP as well as elsewhere to provide the gains in engine materials that are required for economic viability and environmental acceptability. Also, HITEMP is closely coordinated with the joint DOD/NASA Integrated High Performance Turbine Engine Technology Program (IHPTET), and new materials from HITEMP may be utilized in future military applications.

Numerical Propulsion System Simulation



Validated models from basic disciplines

Integrated interdisciplinary analysis and assessment of propulsion systems

- Performance
- Stability
- Durability
- Life

High performance computing
- Processing
- Networks
- Displays

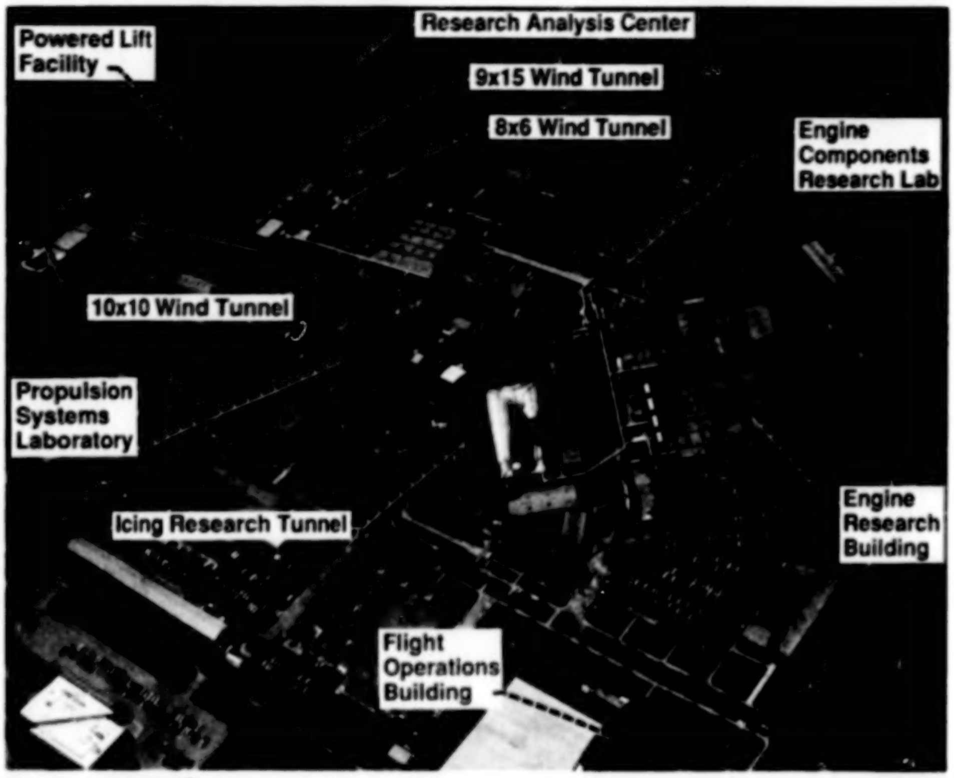
A numerical "test cell" for aerospace propulsion systems

CD-91-54020

The cost of implementing new technology in aerospace propulsion systems is becoming extremely expensive. One of the major contributors to the high cost is the need to perform many large scale system tests. Extensive testing is used to capture the complex interactions among the multiple disciplines and the multiple components inherent in complex systems. The objective of the Numerical Propulsion System Simulation (NPSS) is to provide insight into these complex interactions through computational simulations. This will allow for comprehensive evaluation of new concepts earlier in the design phase before a commitment to hardware is made. It will also allow for rapid assessment of field-related problems.

The NPSS approach means the coupling of disciplines and components computationally to determine system attributes such as performance, reliability, stability, and life. Since these system attributes have traditionally been obtained in the test cell, NPSS is referred to as the "numerical test cell". Such an integrated interdisciplinary system analysis requires advancements in the following technologies: (1) interdisiplinary analyses to couple the relevant disciplines such as aerodynamics, structures, heat transfer, chemistry, materials, controls; (2) integrated system analysis to couple subsystems, components, and subcomponents at an appropriate level of detail; (3) high performance computing platforms composed of a variety of architectures, including massively parallel processors, to provide the required computing speed and memory; and (4) user-friendly simulation environment.

Aeropropulsion Facilities


Powered Lift Facility
Research Analysis Center
9x15 Wind Tunnel
8x6 Wind Tunnel
Engine Components Research Lab
10x10 Wind Tunnel
Propulsion Systems Laboratory
Icing Research Tunnel
Engine Research Building
Flight Operations Building

CD-91-54021

ORIGINAL PAGE IS
OF POOR QUALITY

Both experimental and computational facilities support our aeropropulsion research. Lewis wind tunnels offer a broad range of capabilities from near-static to supersonic. Unique capabilities at low speed are provided by the 9x15 and IRT. The continuous running 8x6 and 10x10 tunnels can test operating propulsion systems over a range of Mach numbers from 0.4 to 3.5. During the next several years, the Hypersonic Test Facility (Mach 5 to 7) at Plum Brook and a 21-in. Hypersonic Wind Tunnel (Mach 4 to 11), obtained from JPL, will be reactivated.

In the area of engine test cells, the Propulsion Systems Laboratory consists of two chambers capable of testing large-scale engines to Mach 3.5 and 70,000 ft conditions. A recent modification has enabled direct-connect testing at Mach 6 and 120,000 ft conditions with reduced mass flow. The Engine Components Research Laboratory provides for full-scale sea-level testing of turboshaft engines. Built primarily to support STOVL research, the Powered Lift Facility provides capabilities to test propulsive lift devices.

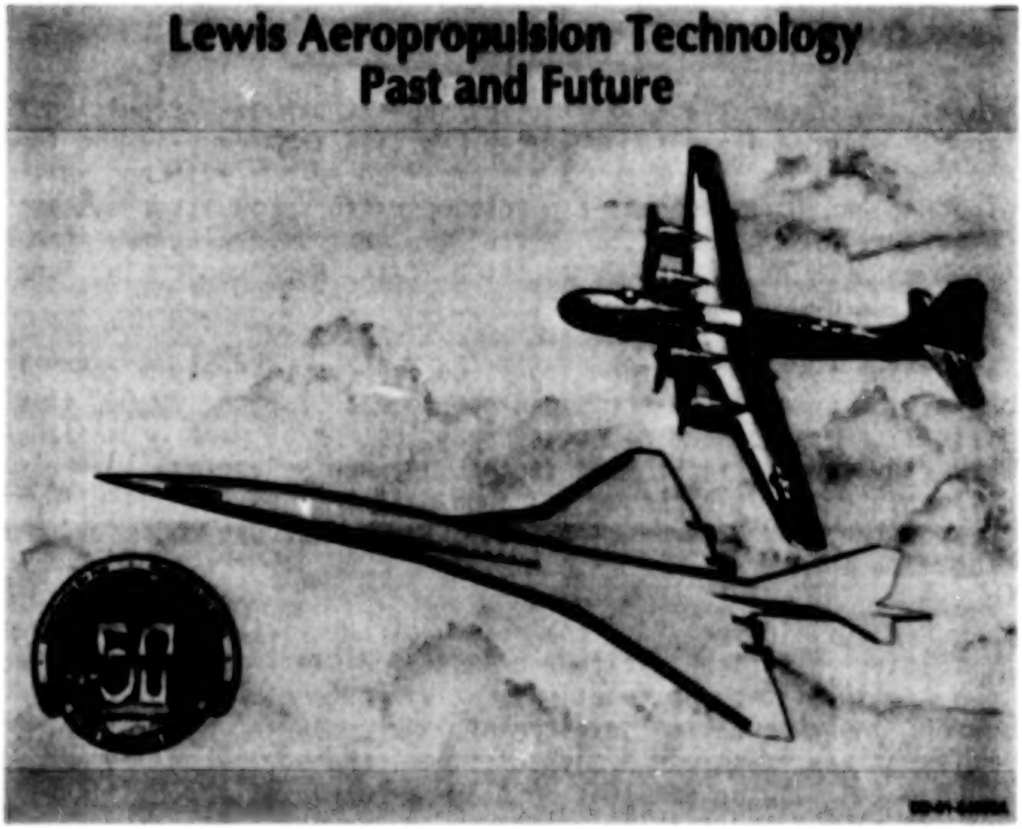
The Engine Research Building provides compressed air at various temperatures, pressures, and humidities as well as altitude exhaust and electrical power to some 60 test cells. A multiplicity of functions can be performed therein including fundamental research, instrumentation development, and component evaluation. Computational facilities at Lewis include a class 7 supercomputer, satellite access to the National Aerodynamic Simulator (NAS) at NASA Ames, top-level scientific workstations, and local-area networks. And finally, although we are not a flight research center, Lewis does have aircraft that are used to augment ground-based testing.

CONCLUDING REMARKS

The past half century at NACA/NASA Lewis has been one of great accomplishment for aeropropulsion technology. From fixing the B-29 engine problems to validating the advanced turboprop, we have contributed to expanding the aeropropulsion operating envelope while improving fuel efficiency, environmental acceptability, and flight safety. The award of the Collier Trophy, the most coveted award in the aerospace field, suggests that our strategy of emphasizing high-payoff high-risk innovative aeropropulsion research and working closely with our industrial and academic partners is having success.

We certainly remain bullish on the future for aeropropulsion research at Lewis. Improvements in propulsion technology historically have provided a major share of aircraft performance improvements, and propulsion advances will be even more critical to the development of future aircraft. So we plan to continue to play a strong leadership role in advancing technology for future aircraft engines. More details of our current and planned efforts are described in the following sessions of this conference.

In summary, we are committed to maintaining a leading-edge research program, a talented staff, and world-class facilities in order to challenge the future and provide the technology advancements for 21st century aeropropulsion.



Lewis Aeropropulsion Technology
Past and Future

N91-20088

OVERVIEW OF SUPERSONIC CRUISE PROPULSION RESEARCH

R.J. Shaw
NASA Lewis Research Center
Cleveland, Ohio

Significant advances in propulsion performance are required if supersonic transport vehicles are to become an important part of the 21st century international air transportation system. The objective of the NASA Lewis Supersonic Cruise propulsion research is to provide the critical propulsion technologies to the industry in a timely fashion to contribute to the design of economically viable and environmentally acceptable high-speed civil transport (HSCT).

Supersonic Cruise

Objective:

Provide enabling propulsion technology for economically viable and environmentally compatible supersonic transports

CD-91-54379

Significant advances in propulsion performance are required if supersonic transport vehicles are to become an important part of the 21st century international air transportation system. The objective of the NASA Lewis Supersonic Cruise propulsion research is to provide the critical propulsion technologies to the industry in a timely fashion to contribute to the design of economically viable and environmentally acceptable high-speed civil transport (HSCT).

Supersonic Cruise Propulsion Research Areas of Emphasis

- **High-Speed Research Program**

  Develop technology solutions to barrier issues of environmental compatibility and economic viability associated with IOC 2005-2008 High-Speed Civil Transport (HSCT)

- **Supersonic Through-Flow Technology**

  Demonstrate viability of supersonic cruise propulsion system based on a supersonic through-flow fan

CD-91-54379

NASA's Supersonic Cruise propulsion research has both a near-term and far-term emphasis. In the near term, the emphasis is on finding viable technology solutions to the barrier issues of environmental compatibility and economic viability. The goal is to have these technologies in place by the end of the decade to allow industry to make a formal "go/no go" decision on producing an HSCT which would begin service in the 2005 to 2008 time period. Efforts are also underway to look at alternate propulsion system concepts that may be attractive candidates for future generation HSCT's. The current long-term emphasis is to develop the technology base for a supersonic through-flow fan propulsion system. This paper will present an overview of both aspects of the Supersonic Cruise research program.

Supersonic Cruise Propulsion Research

System Studies
- Variable cycle engines
- Supersonic through-flow fan engines

Focused Technology Research
- Environmental concerns
  - NO_x emissions
  - Community noise
- Supersonic through-flow turbomachinery
  - Fan stage
  - Inlets
- Conventional supersonic inlets
- Enabling propulsion materials

Fundamental R & T
- Analysis codes
  - Inlet
  - Supersonic fan
  - Combustion
  - Nozzles (aero/acoustic)
- Fluid physics
  - Shear flow control
  - Fundamental aeroacoustics

CD-91-54380

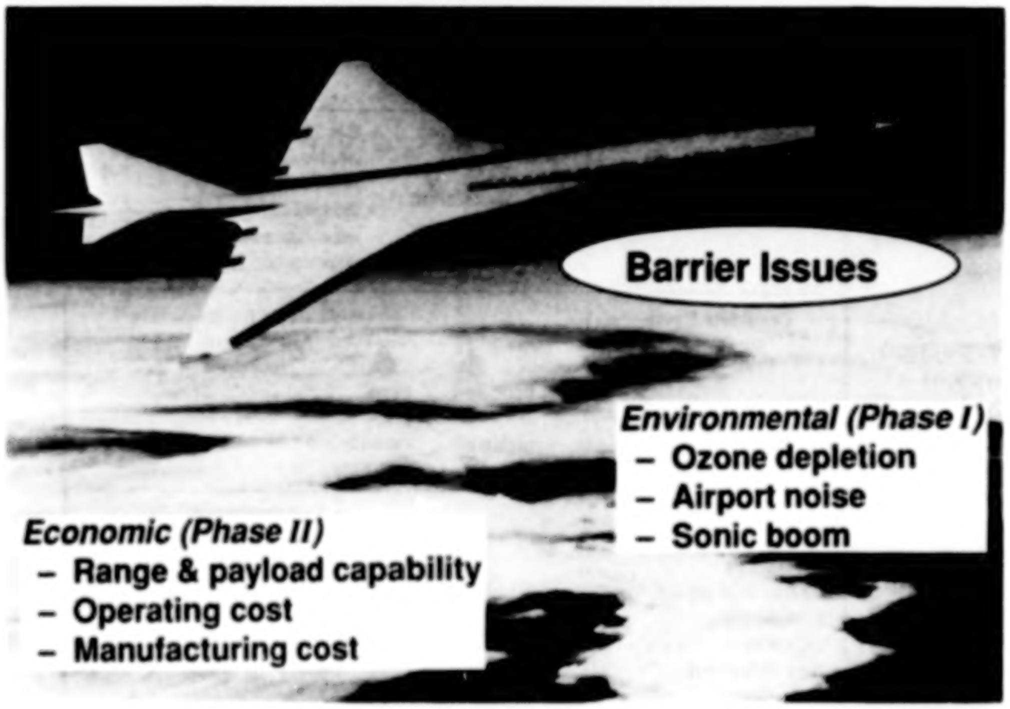
The NASA Supersonic Cruise propulsion research is composed of three major components:

1. System Studies evaluate various candidate propulsion cycles and develop technology roadmaps for the most promising systems.

2. Focused Technology Research represents the dedicated efforts to carry the propulsion research to the required level of development and validation.

3. Fundamental Research and Technology represents the basic efforts required to achieve the goals of the focused technology research.

High-Speed Research Program



Barrier Issues

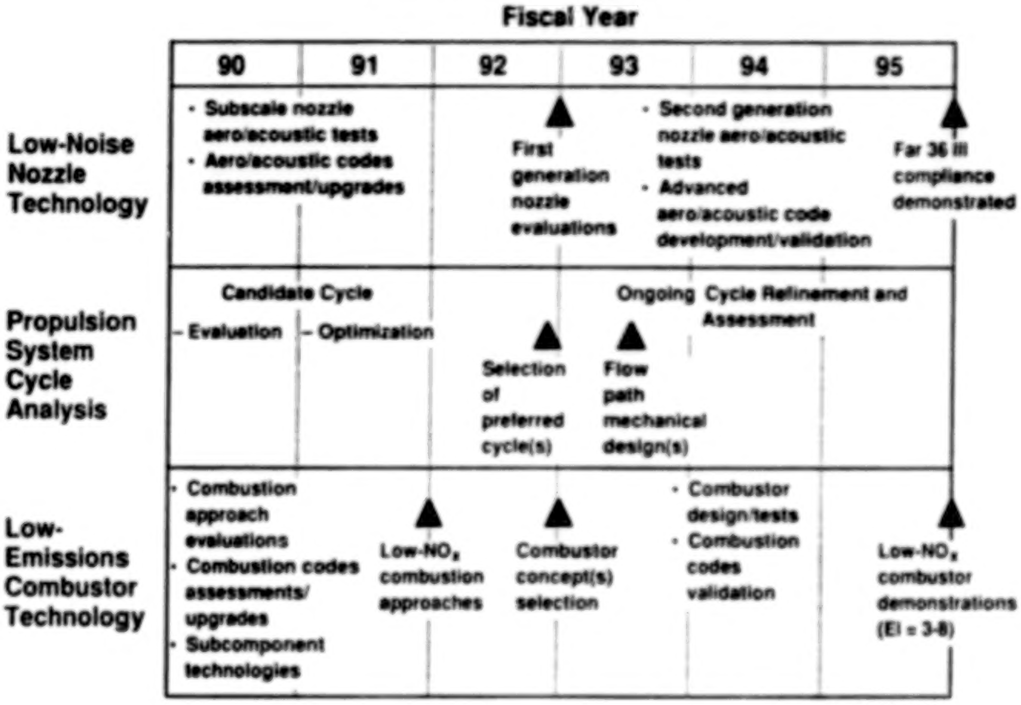
Environmental (Phase I)
- **Ozone depletion**
- **Airport noise**
- **Sonic boom**

Economic (Phase II)
- **Range & payload capability**
- **Operating cost**
- **Manufacturing cost**

CD-91-54381

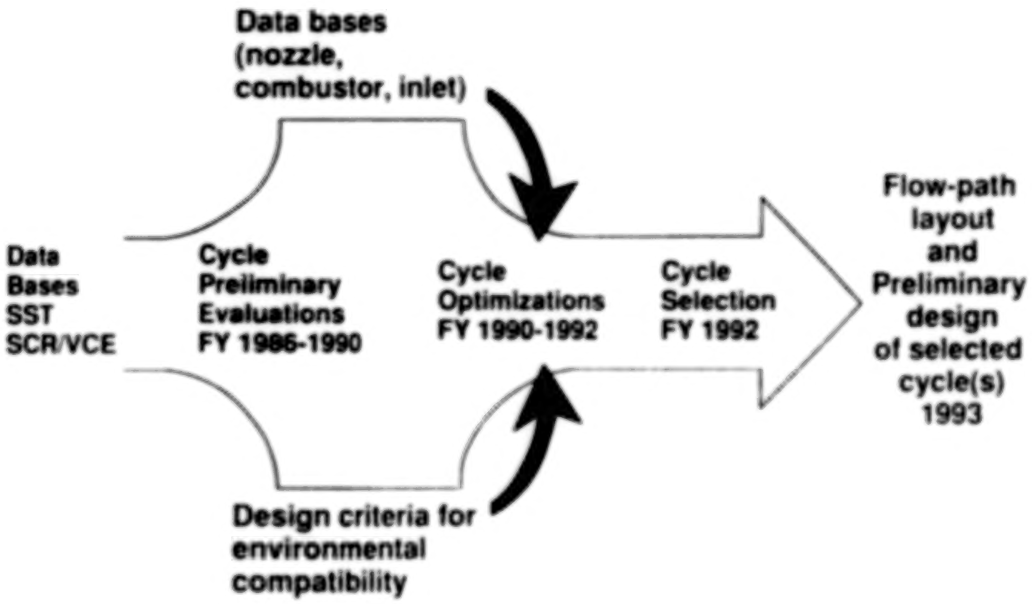
The NASA Phase I High-Speed Research Program (HSRP) emphasizes solutions to the critical environmental barrier issues associated with any future HSCT aircraft. Two of these barrier issues - atmospheric ozone depletion and community noise - are primarily propulsion issues and are addressed in the Lewis portion of HSRP. The critical economical viability issues will be the emphasis of a proposed Phase II HSRP, which could be initiated as early as FY 1992.

NASA Lewis Involvement in High-Speed Research Program


Fiscal Year
90
91
92
93
94
95
Low-Noise Nozzle Technology
· Subscale nozzle aero/acoustic tests
· Aero/acoustic codes assessment/upgrades
First generation nozzle evaluations
· Second generation nozzle aero/acoustic tests
· Advanced aero/acoustic code development/validation
Far 36 III compliance demonstrated
Propulsion System Cycle Analysis
Candidate Cycle
– Evaluation
– Optimization
Selection of preferred cycle(s)
Flow path mechanical design(s)
Ongoing Cycle Refinement and Assessment
Low-Emissions Combustor Technology
· Combustion approach evaluations
· Combustion codes assessments/upgrades
· Subcomponent technologies
Low-NO$_x$ combustion approaches
Combustor concept(s) selection
· Combustor design/tests
· Combustion codes validation
Low-NO$_x$ combustor demonstrations (EI = 3-8)
CD-91-54382

The three elements which comprise the Lewis portion of HSRP Phase I are shown with the major program milestones indicated. Low-noise nozzle and low-NO$_x$ combustor technology efforts will result in scale model demonstrations of component concepts which meet environmental and performance requirements. Concurrent with these component technology efforts are ongoing propulsion cycle evaluations. These evaluations will incorporate the latest component technology data and airframe installation evaluations resulting in the selection of the preferred cycle(s), flow-path layouts, and preliminary mechanical designs. The selection of the preferred cycle(s) will be based on the considerations of environmental compatibility and economic viability. The following charts describe in more detail these three elements of the HSRP Phase I program.

HSCT Propulsion Cycle Evaluation and Selection Process


Data bases
(nozzle,
combustor, inlet)
Data
Bases
SST
SCR/VCE
Cycle
Preliminary
Evaluations
FY 1986-1990
Cycle
Optimizations
FY 1990-1992
Cycle
Selection
FY 1992
Flow-path
layout
and
Preliminary
design
of selected
cycle(s)
1993
Design criteria for
environmental
compatibility

CD-91-54383

The propulsion cycle evaluation process is depicted in this figure. The process began with the information from the U.S. Supersonic Transport (SST) and NASA Supersonic Cruise Research/Variable Cycle Engine (SCR/VCE) programs. From propulsion cycles considered during these programs and others more recently conceived, potentially attractive cycles were chosen. They are being evaluated and compared on the bases of environmental compatibility and economic viability.

The factors used to compare and evaluate the candidate cycles include aircraft takeoff gross weight, fuel burn, emissions level and atmospheric impact, and takeoff and landing noise levels.

The cycle evaluation process is iterative and is closely coupled to the propulsion and airframe component technology efforts underway in HSRP. Specifically, as performance results for the various component concepts are acquired, they are factored into the cycle performance predictions and a new round of cycle comparisons is conducted. Other critical input factors include the evolving design criteria for environmental compatibility. The cycle evaluation process is directly influenced by the ongoing ozone impact assessments being conducted in another element of HSRP and by FAA noise certification rules being developed for any future HSCT's.

As the figure indicates, the cycle evaluation process will be conducted through FY 1992. At that time it is envisioned that adequate information will be available to select the cycle(s) which will focus the remainder of the HSRP component and systems integration technology research. For each of the cycles selected, flow-path layouts and preliminary mechanical designs will be completed.

Candidate HSCT Propulsion Concepts

Turbine Bypass Engine

Key Features:

- Simplicity
- Low cruise combustor temperature

Mixed Flow Afterburning Turbofan

Key Features:

- Low jet velocity
- Good subsonic SFC

Variable Cycle Engine

Key Features:

- Variable bypass
- Good subsonic SFC

"F" Engine

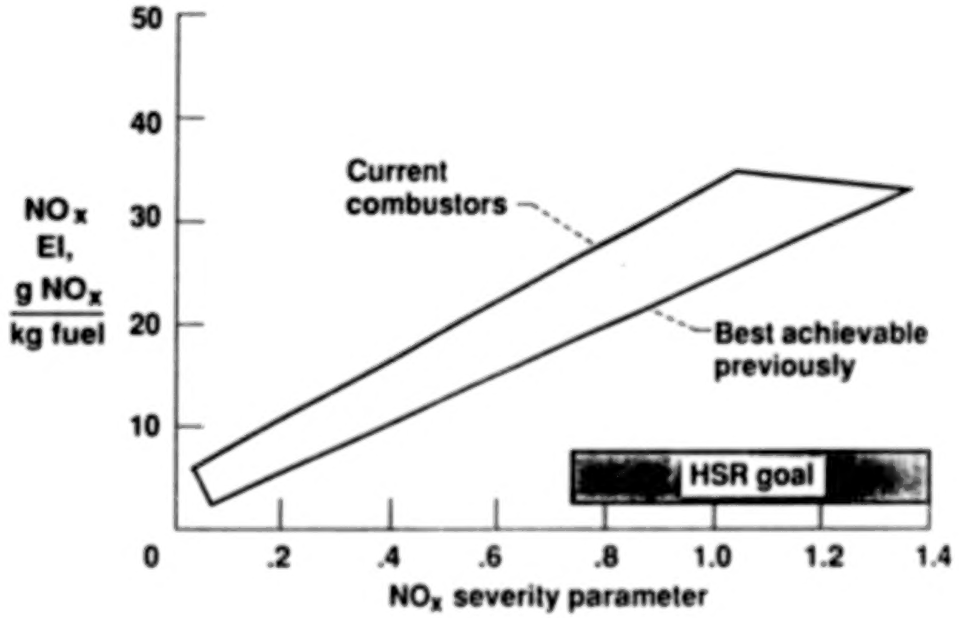
Key Features:

- Variable bypass
- Good subsonic SFC
- Low jet noise

CD-91-54384

Currently the propulsion cycle evaluation process is concentrating on the four cycles shown. The key features for each of the cycles are indicated.

HSR NO_x Emissions Challenge



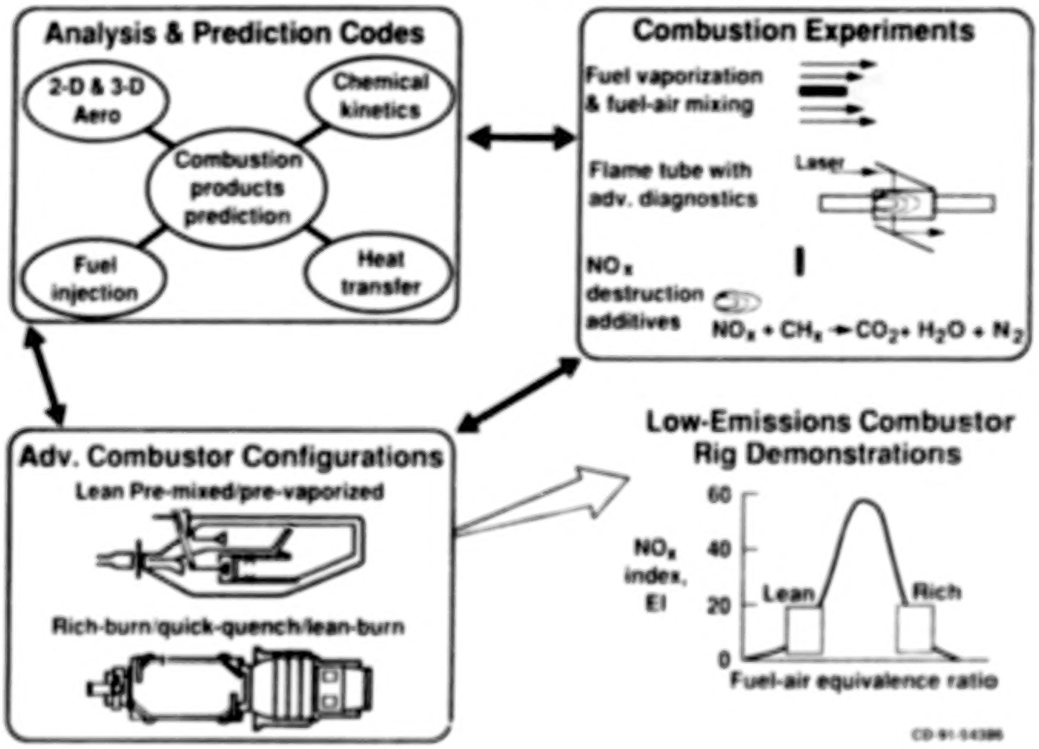
CD-91-54385

The HSR NO_x emissions challenge is shown in this figure. Initial two-dimensional atmospheric impact studies suggest that ultra low NO_x combustor technology will be required if no adverse impact on the ozone layer is to occur. The standard term for expressing NO_x emissions levels is the emissions index (EI), defined as

$$EI = \frac{g\ of\ equivalent\ NO_2\ produced}{kg\ of\ fuel\ burned}$$

These ultra-low NO_x levels would have EI's in the range of 3 to 8. The figure shows the emissions parameter as a function of a severity parameter, which is itself a function of combustor pressure and temperature levels. The HSRP goal is compared to the performance of current in-the-fleet combustors and to performance levels demonstrated in the NASA/Industry Experimental Clean Combustor Program.

Low-Emissions Combustor Technology Elements


Analysis & Prediction Codes
2-D & 3-D Aero
Chemical kinetics
Combustion products prediction
Fuel injection
Heat transfer
Combustion Experiments
Fuel vaporization & fuel-air mixing
Flame tube with adv. diagnostics
Laser
NO$_x$ destruction additives
NO$_x$ + CH$_x$ → CO$_2$ + H$_2$O + N$_2$
Adv. Combustor Configurations
Lean Pre-mixed/pre-vaporized
Rich-burn/quick-quench/lean-burn
Low-Emissions Combustor Rig Demonstrations
NO$_x$ index, EI
60
40
20
0
Lean
Rich
Fuel-air equivalence ratio

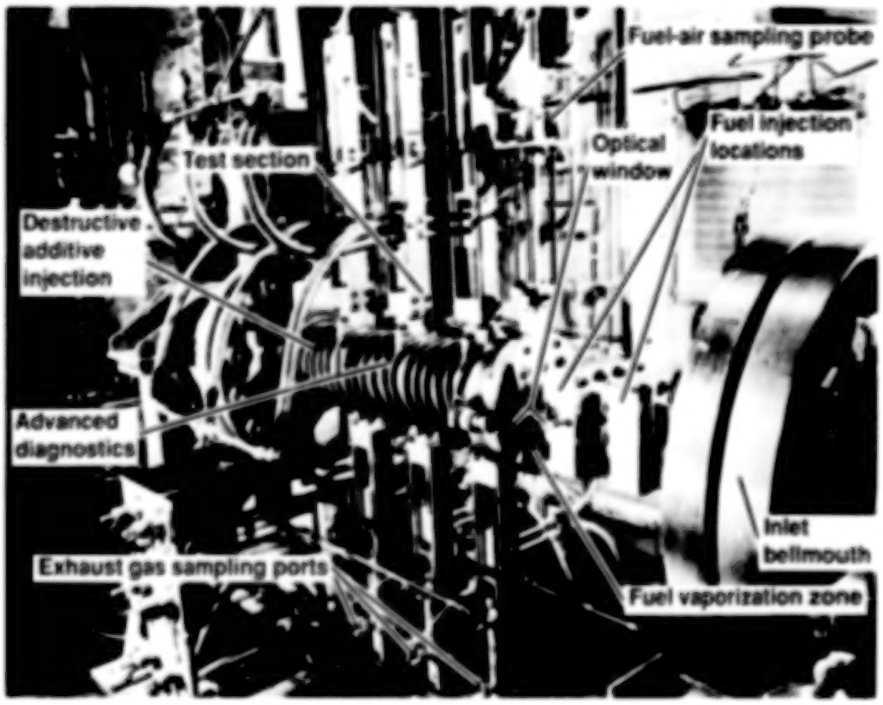
The major elements of the low emissions combustor technology portion of HSRP are shown. Initially, emphasis will be on the development and validation of the computer analyses to predict the details of the combustion process within candidate combustor configurations. Also, laboratory experiments will be conducted to evaluate candidate low-NO$_x$ combustion approaches. These laboratory tests will be used in conjunction with advanced diagnostics to develop a comprehensive combustion code validation data base.

These experimental data bases and the analytical prediction codes will form the basis for the conceptual design of candidate low-NO$_x$ combustors. The deliverable of this element of HSRP will be the demonstration of ultra-low-NO$_x$ combustor configurations in rig demonstrations.

Currently, two combustor concepts appear to hold promise for meeting the HSRP emissions goal of EI = 3 to 8: the lean-premixed-prevaporized (LPP) and the rich-burn/quick-quench/lean-burn (RQL). The key to achieving ultra-low-NO$_x$ production levels is to accomplish all burning away from stoichiometric conditions. The LPP concept features burning at lean fuel-air conditions. The RQL concept requires two stages of burning. The first stage burning is conducted in a fuel rich environment. The transition from rich to lean burning is accomplished through an introduction of quench air between the two stages. The quench air must be introduced into the combustion stream so that mixing occurs rapidly and uniformly such that no localized burning zones occur at close to stoichiometric conditions, which would significantly increase the NO$_x$ produced.

HSR Square Flame Tube Rig

Lean-Premixed-Prevaporized Combustion (LPP)

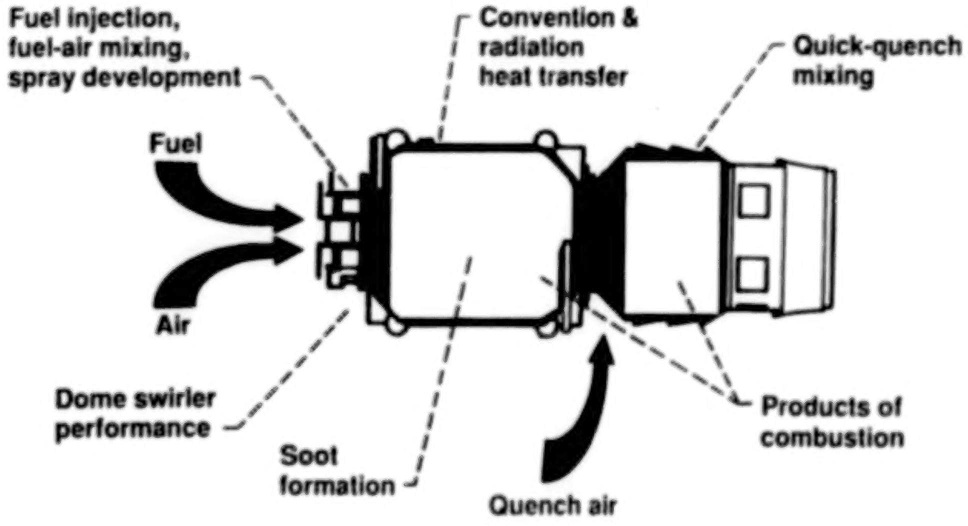

Fuel-air sampling probe
Fuel injection locations
Optical window
Test section
Destructive additive injection
Advanced diagnostics
Inlet bellmouth
Exhaust gas sampling ports
Fuel vaporization zone

ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-54387

The lean-premixed-prevaporized (LPP) flame tube rig is shown with the key facility characteristics indicated. A similar rig will shortly be operational to evaluate the RQL combustion concept.

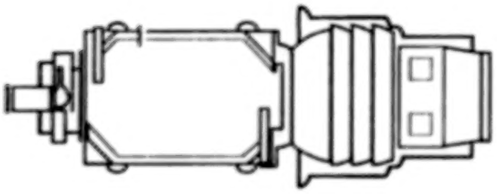
Computer Modeling Requirements for HSCT Combustors

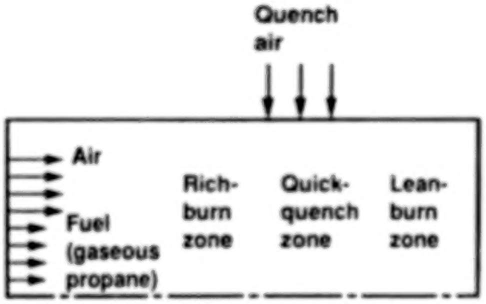

Fuel injection,
fuel-air mixing,
spray development
Convention &
radiation
heat transfer
Quick-quench
mixing
Fuel
Air
Dome swirler
performance
Soot
formation
Quench air
Products of
combustion

CD-91-54388

The computer modeling requirements for HSCT combustors are shown here. To be a useful analysis and design tool, combustor modeling codes must be able to predict the critical physics of combustor flow fields as depicted.

Low NO_x Combustor Analysis





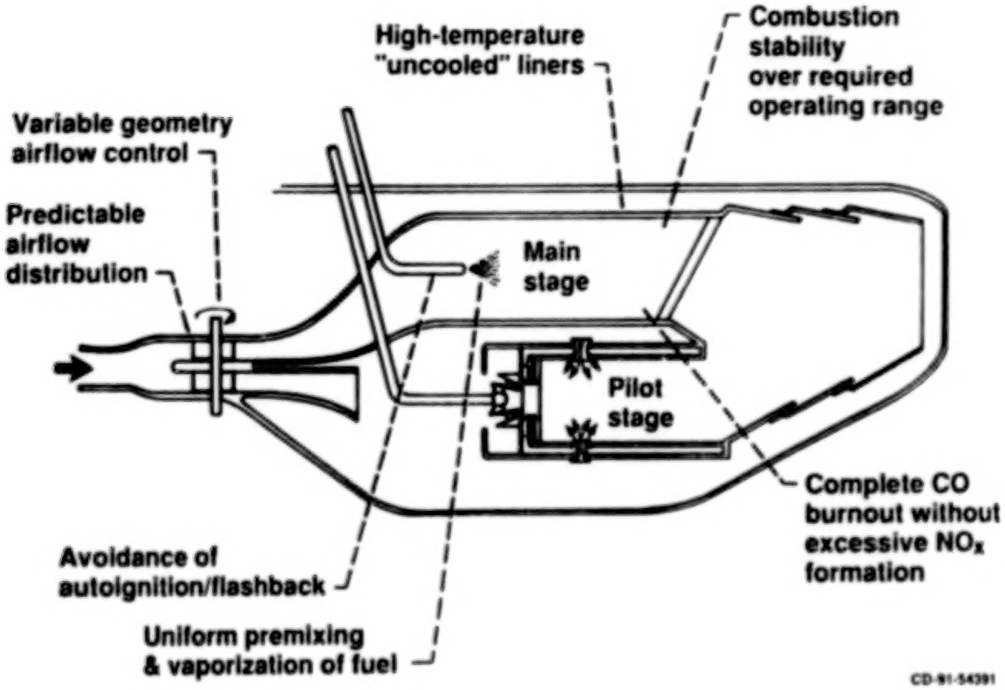
Simplified Combustor Model

Predicted Temperature Contours

CD-91-54389

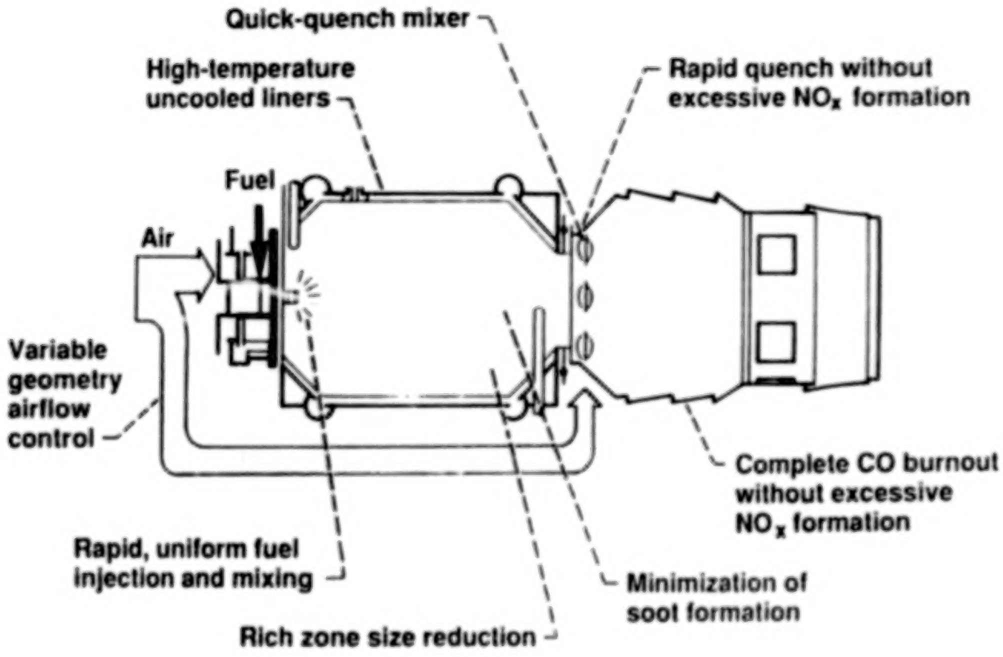
The approach being followed in HSRP is to take existing available computer codes and upgrade, modify, and validate them for HSRP applications. The two codes being emphasized to analyze low NO_x combustors are the KIVA II code developed by Los Alamos and the Lewis Research Center Code (LeRC 3D) developed as part of the NASA rotary engine program. This figure depicts an early attempt to model the RQL combustion process wherein the geometry being considered is an idealized axisymmetric configuration. Temperature contours are shown although velocity and pressure contours are also predicted as are important species such as NOx concentration contours. Both codes are undergoing extensive upgrades to include improved grid generation, turbulence modeling and chemical kinetics. The codes are also being applied to the critical combustor subcomponent problems such as dome swirlers, fuel injectors, and quick-quench mixers. The deliverable from these efforts will be a series of codes sufficiently calibrated and validated against experimental data that they can be used as a part of the actual combustor design process.

Technology Issues – LPP Combustors


High-temperature "uncooled" liners
Combustion stability over required operating range
Variable geometry airflow control
Predictable airflow distribution
Main stage
Pilot stage
Complete CO burnout without excessive NO_x formation
Avoidance of autoignition/flashback
Uniform premixing & vaporization of fuel
CD-91-54391

This figure indicates some of the critical technology challenges that must be overcome before ultra-low-NO_x LPP combustors could be designed.

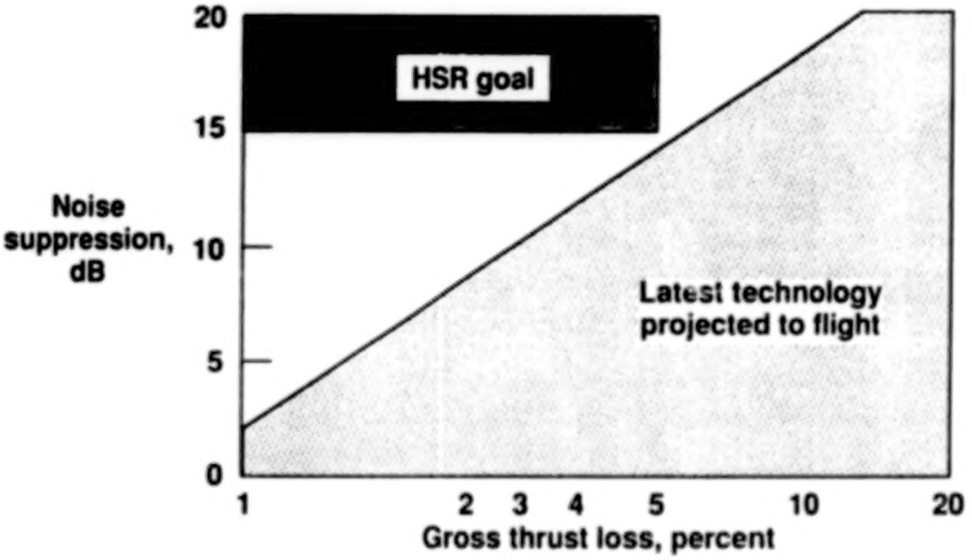
Technology Issues – RQL Combustors


Quick-quench mixer
High-temperature uncooled liners
Rapid quench without excessive NO_x formation
Fuel
Air
Variable geometry airflow control
Complete CO burnout without excessive NO_x formation
Rapid, uniform fuel injection and mixing
Minimization of soot formation
Rich zone size reduction

CD-91-54392

This figure indicates some of the critical technology challenges that must be overcome before ultra-low-NO_x RQL combustors could be designed. HSRP combined experimental and analytical efforts will develop the required combustor subcomponent technologies for LLP and RQL combustors and incorporate them as required in the combustor rig demonstrations at the conclusion of the program.

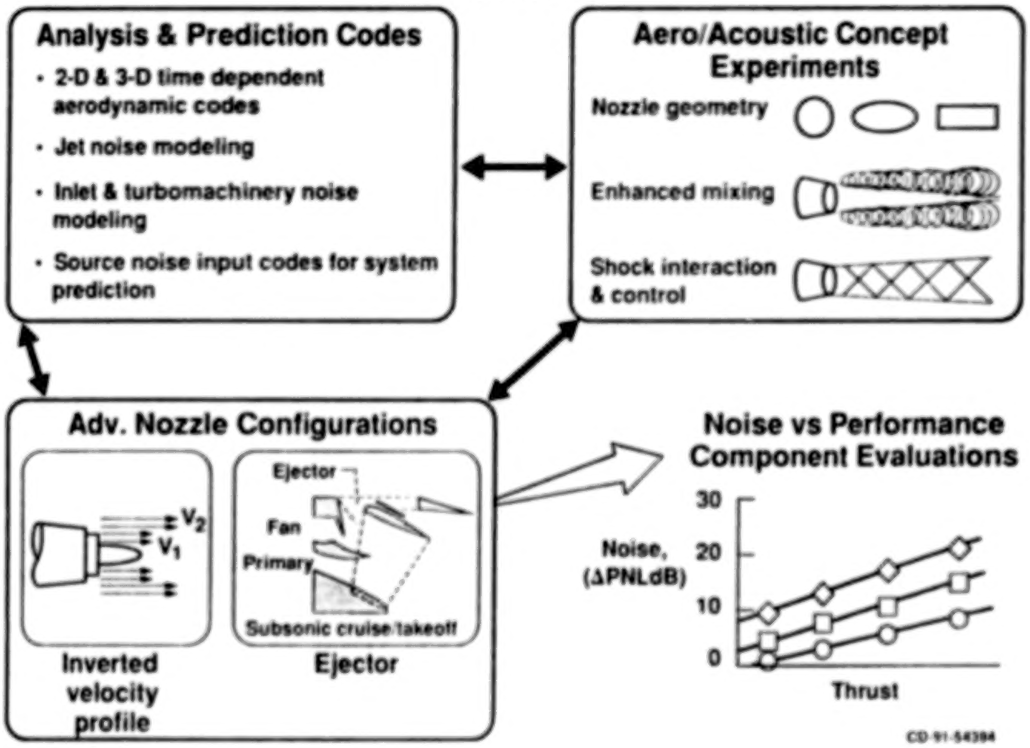
HSCT Source Noise Challenge



CD-91-54393

The HSCT source noise challenge is illustrated in this figure. The jet exhaust noise levels at takeoff and landing conditions must be reduced by 15 to 20 db relative to reference conic nozzle levels before any future HSCT can hope to have noise levels below FAA noise regulation limits. At the same time, the nozzle aerodynamic performance levels must be kept high if vehicle overall mission performance goals are to be met. This combined acoustic-aerodynamic challenge is often expressed as a ratio of decibel noise reduction to resultant percent thrust loss. For a viable HSCT design this ratio should be in the neighborhood of 4:1. As this figure shows, current technology would yield a nozzle design with a ratio of no better than 2:1.

Low-Noise Nozzle Technology Elements



Analysis & Prediction Codes

- 2-D & 3-D time dependent aerodynamic codes
- Jet noise modeling
- Inlet & turbomachinery noise modeling
- Source noise input codes for system prediction

Aero/Acoustic Concept Experiments

Nozzle geometry

Enhanced mixing

Shock interaction & control

Adv. Nozzle Configurations

Inverted velocity profile

Ejector

Noise vs Performance Component Evaluations

The major elements of the source noise portion of HSRP are shown in this figure. Much like the low NO_x combustor area, heavy emphases are being placed in the first years of HSRP on computer code development and validation and on subscale experiments to evaluate potentially attractive nozzle concepts. The emphases regarding the codes is again on applying available solvers for both nozzle aerodynamic flows and for the acoustic signatures of the various configurations. The laboratory experiments and computer code developments and the insights they provide as to the governing fluid physics will be key inputs to the development of advanced nozzle configurations that will meet the HSRP goals, both for aerodynamic performance and acoustic suppression.

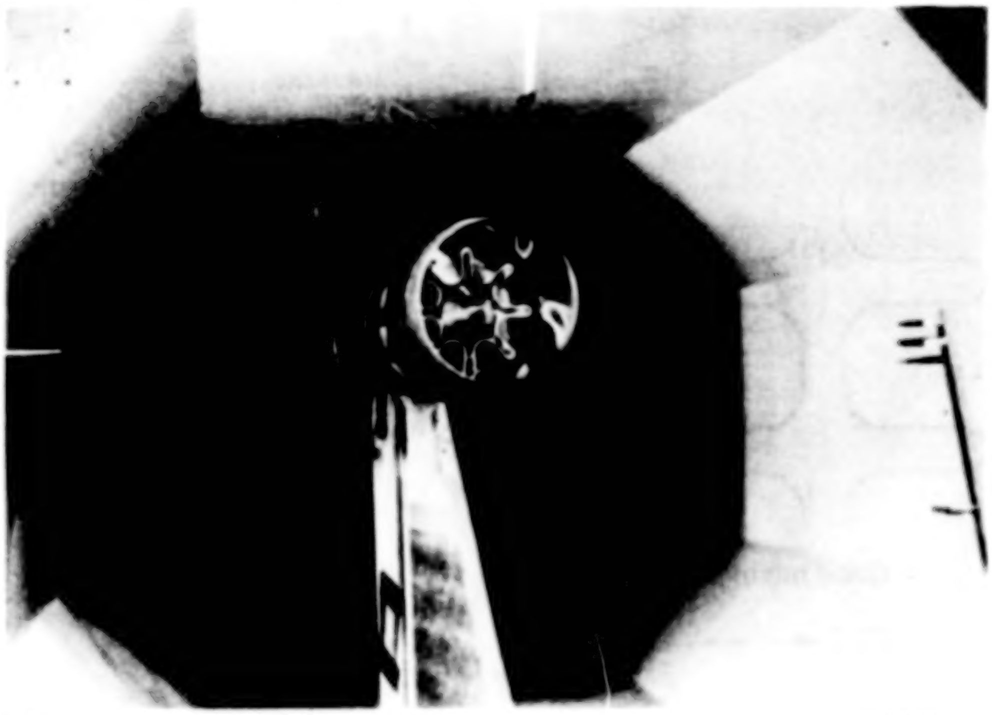
Two-Dimensional Mixer-Ejector Nozzle in NASA Lewis 9x15 LSWT

CD-91-54390

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This figure shows a two-dimensional model tested in the NASA Lewis 9 x 15 Low-Speed, Anechoic Wind Tunnel (LSWT) to evaluate a Pratt and Whitney mixer-ejector nozzle concept. Aerodynamic flow field surveys as well as near-field acoustic signatures were made for a series of configurations.

Pratt and Whitney Axisymmetric Mixer-Ejector Nozzle in Boeing Low-Speed Aeroacoustic Facility

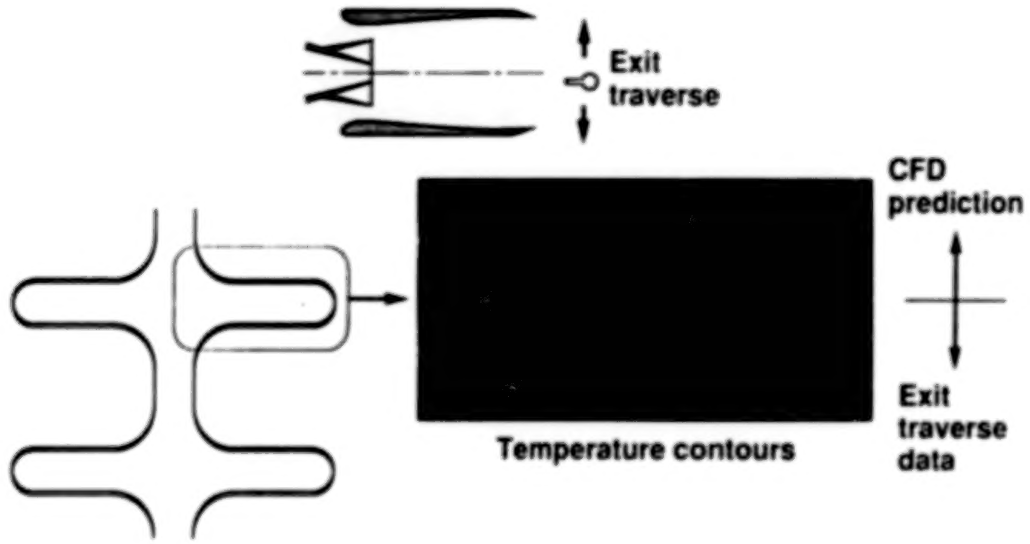


ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-54395

The data from the two-dimensional concept model test in the 9x15 LSWT were used to help in designing an axisymmetric mixer-ejector nozzle configuration. This model was tested in the Boeing Low-Speed Aeroacoustic Facility (LSAF) to measure nozzle aerodynamic performance and acoustic signatures.

CFD/Translating Probe Comparison and Mixing

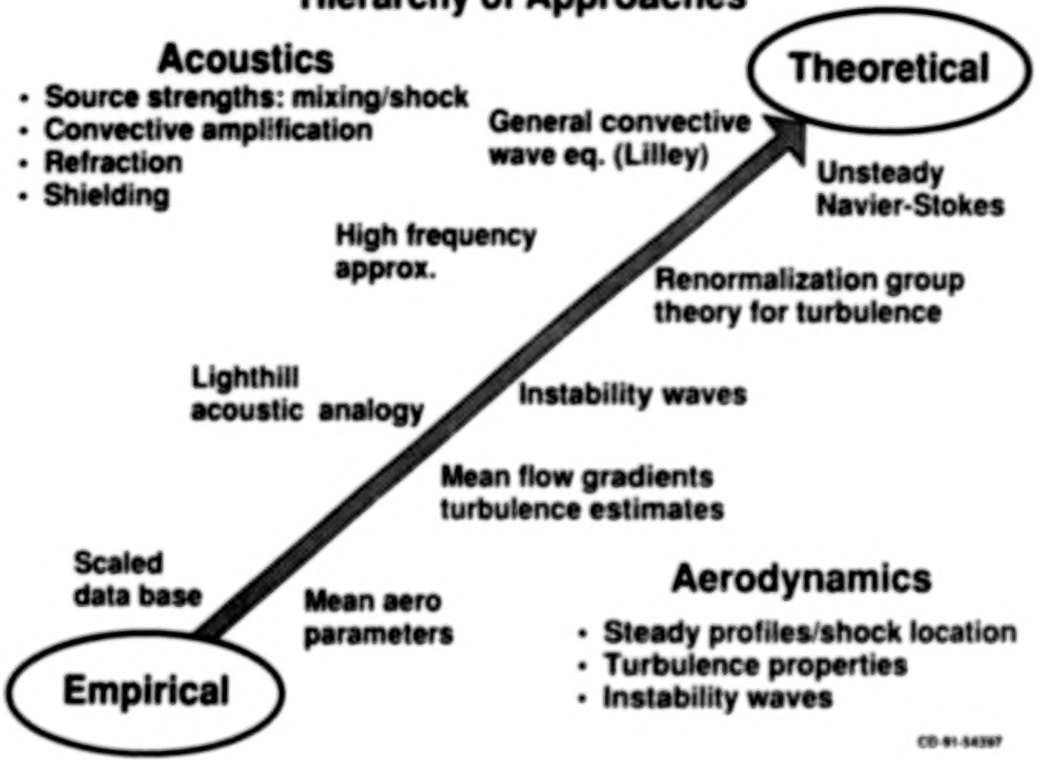

Exit
traverse
CFD
prediction
Exit
traverse
data
Temperature contours

- Good mixing
- CFD ←—→ Exit traverse:
  Excellent agreement

CD-91-54396

The data from both the two-dimensional and axisymmetric model tests are being compared with Navier-Stokes solver predictions of the nozzle aerodynamic flowfields. This figure shows a comparison of measured and predicted total temperature profiles at the exit of one of the two-dimensional model configurations. The comparison is judged to be excellent. Results such as these suggest the important role computer codes will have in the analysis of nozzle complex aerodynamic flow fields.

Jet Noise Aeroacoustic Analysis
Hierarchy of Approaches



Acoustics
- Source strengths: mixing/shock
- Convective amplification
- Refraction
- Shielding

Theoretical

General convective wave eq. (Lilley)

Unsteady Navier-Stokes

High frequency approx.

Renormalization group theory for turbulence

Lighthill acoustic analogy

Instability waves

Mean flow gradients turbulence estimates

Scaled data base

Mean aero parameters

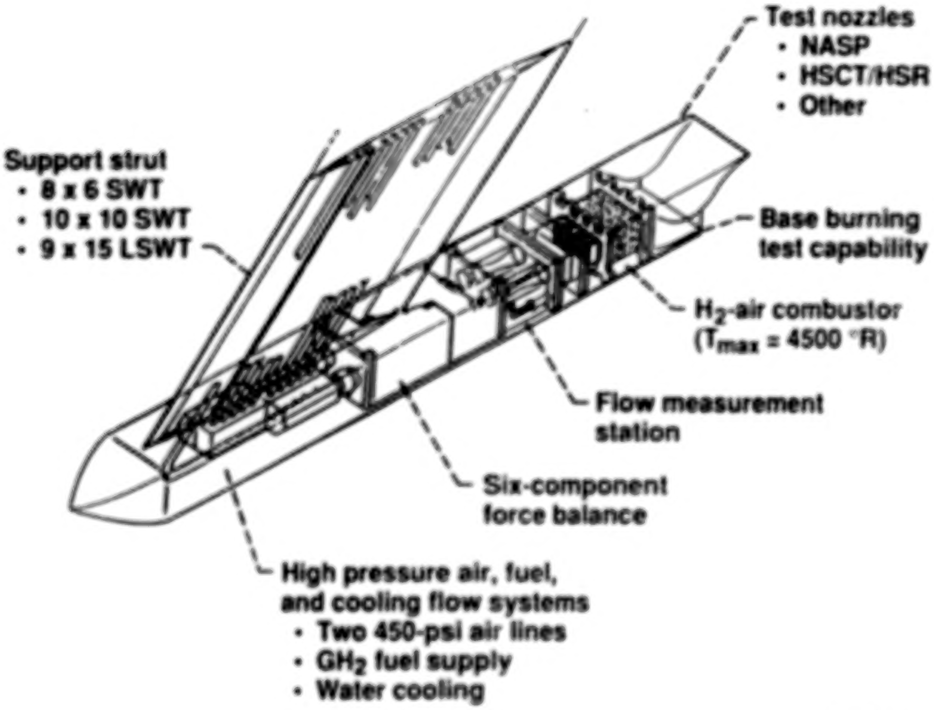
Empirical

Aerodynamics
- Steady profiles/shock location
- Turbulence properties
- Instability waves

CD-91-54397

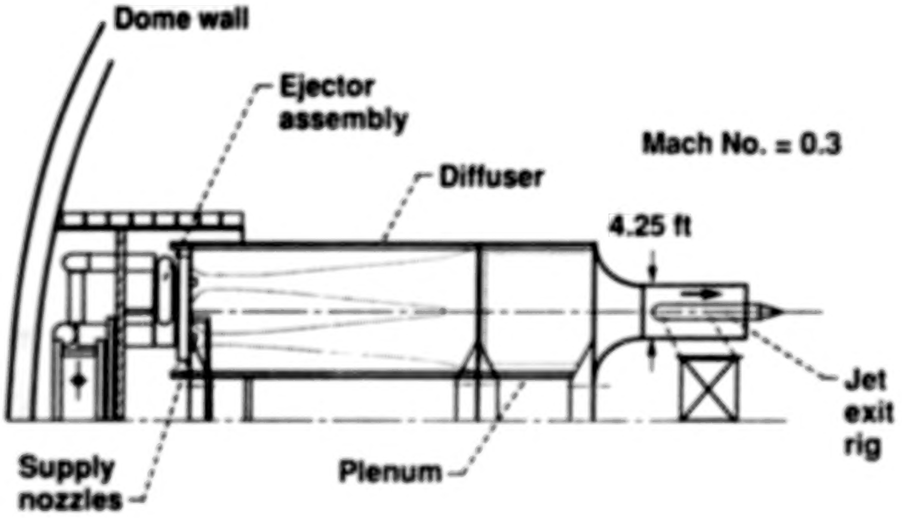
A hierarchy of approaches is being followed to predict HSCT nozzle acoustic signatures. Currently available tools are largely empirical, and these approaches are being updated with the extensive data base being acquired in HSRP. The rapidly increasing computer power available offers the opportunity to develop more fundamentals based prediction schemes for jet noise. Several investigators are pursuing those approaches as part of HSRP.

Jet Exit Rig Details


Test nozzles
• NASP
• HSCT/HSR
• Other
Support strut
• 8 x 6 SWT
• 10 x 10 SWT
• 9 x 15 LSWT
Base burning test capability
H_2-air combustor (T_{max} = 4500 °R)
Flow measurement station
Six-component force balance
High pressure air, fuel, and cooling flow systems
• Two 450-psi air lines
• GH_2 fuel supply
• Water cooling

Advanced experimental test capabilities are being developed as part of HSRP to allow the extensive testing of the many nozzle configurations that will be required to meet the HSRP goals. This figure shows the Lewis Research Center jet exit rig which will be used to test HSRP nozzles for both aerodynamic and acoustic performance. The jet exit rig will be used in all Lewis facilities (8x6 and 10x10 supersonic wind tunnels and the 9x15 low-speed anechoic tunnel) and the NASA Ames 40 x 80 wind tunnel; thus it will be the centerpiece of the experimental portion of the low-noise nozzle technology efforts of HSRP.

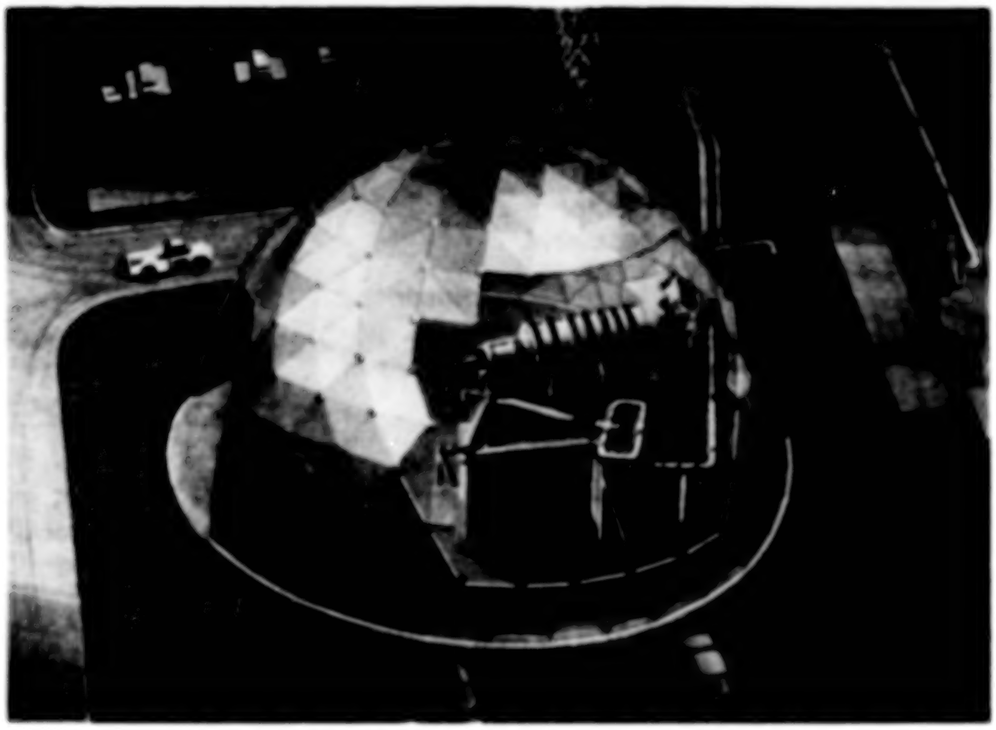
Nozzle Acoustic Test Rig (NATR)


Dome wall
Ejector assembly
Mach No. = 0.3
Diffuser
4.25 ft
Jet exit rig
Supply nozzles
Plenum

CD-91-54399

The extensive amount of nozzle low-speed acoustic testing which will be required in HSRP has resulted in the design and fabrication of a free jet test capability called the nozzle acoustic test rig (NATR) at NASA Lewis. The NATR will simulate forward flight conditions at takeoff and has been designed to be compatible with the jet exit rig already discussed.

Powered Lift and Nozzle Acoustic Rig Test Facilities
(Enclosed by geodesic dome)

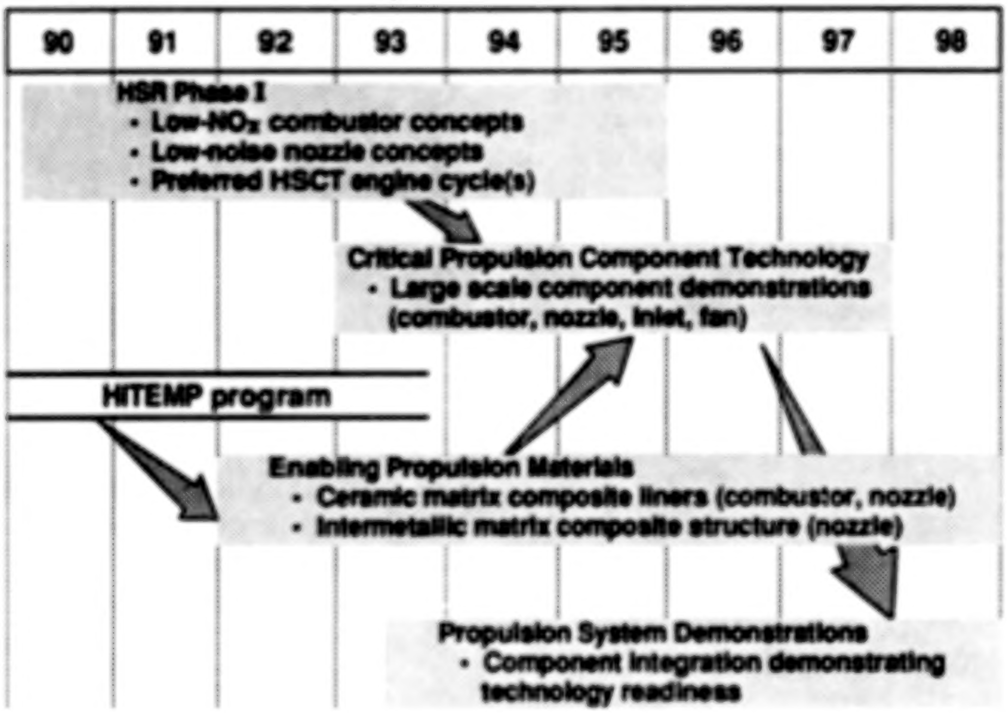


CD-91-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The nozzle acoustic test rig will be located adjacent to the Powered Lift Facility (PLF) at NASA Lewis. In order to alleviate community noise problems and to allow year-round testing, the facilities will be enclosed in a 65-foot-diameter geodesic dome. This figure shows the final version of the design. Appropriate acoustic treatment will be placed in the inside of the dome to allow meaningful HSRP nozzle noise measurements to be made.

NASA High-Speed Research Plan
Propulsion Elements


90
91
92
93
94
95
96
97
98
HSR Phase I
• Low-NO_x combustor concepts
• Low-noise nozzle concepts
• Preferred HSCT engine cycle(s)
Critical Propulsion Component Technology
• Large scale component demonstrations (combustor, nozzle, inlet, fan)
HITEMP program
Enabling Propulsion Materials
• Ceramic matrix composite liners (combustor, nozzle)
• Intermetallic matrix composite structure (nozzle)
Propulsion System Demonstrations
• Component integration demonstrating technology readiness

The roadmap for the propulsion elements of NASA's overall High-Speed Research Program is shown in this figure. HSRP Phase I efforts will result in demonstrations of low-NO_x combustor and low-noise nozzle concepts as well as determination of preferred HSCT propulsion cycles. NASA's HITEMP engine materials program will provide the basis for the development of the advanced composite materials required for the combustor and nozzle components of any future HSCT engine.

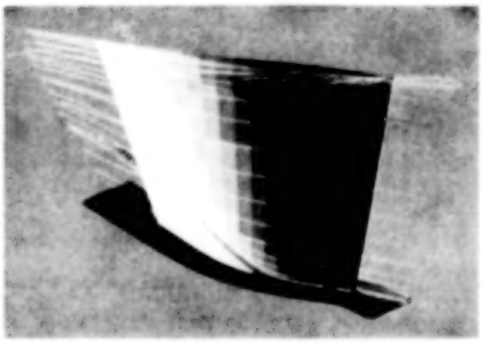
The HSRP Phase I and HITEMP research results will be the inputs to the proposed HSRP Phase II Program currently advocated by NASA. The propulsion elements of HSRP II would demonstrate HSCT propulsion technology readiness initially through large-scale testing of the critical components (inlet, fan, combustor, and nozzle); then these components would be combined with an available core engine in propulsion systems technology demonstrations at both low-speed (takeoff) and high-speed (supersonic cruise) conditions.

The Enabling Propulsion Materials of HSRP II would demonstrate the materials technology readiness through tests of an uncooled ceramic matrix composite (CMC) combustor liner and a nozzle substructure element fabricated from an advanced intermetallic matrix composite (IMC) developed in HSRP II.

Supersonic Through-Flow Technology Program

Objective:

Establish supersonic through-flow technology enabling revolutionary improvement in high-speed aircraft



Focus:

Conduct analytical/experimental research to demonstrate the performance potential of supersonic through-flow compression

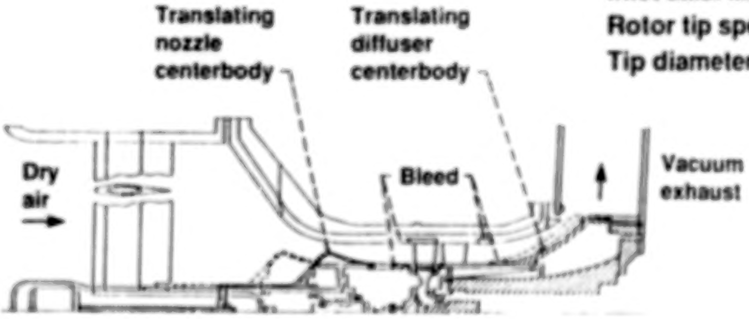
CD-91-54402

The long-term emphasis in the supersonic cruise propulsion research is on examining alternate high performance propulsion system concepts and pursuing the appropriate critical component and system technologies. Currently, the emphasis is on developing the supersonic through-flow technology and, in particular, on demonstrating the viability of the critical components (fan stage, inlets, and nozzle) and eventually system performance and control across the speed range.

Supersonic Through-Flow Program
Baseline Fan

Goals:
Prove concept of a supersonic through-flow fan stage demonstrating subsonic, transition, and supersonic performances; obtain detailed flowfield mapping for code validation

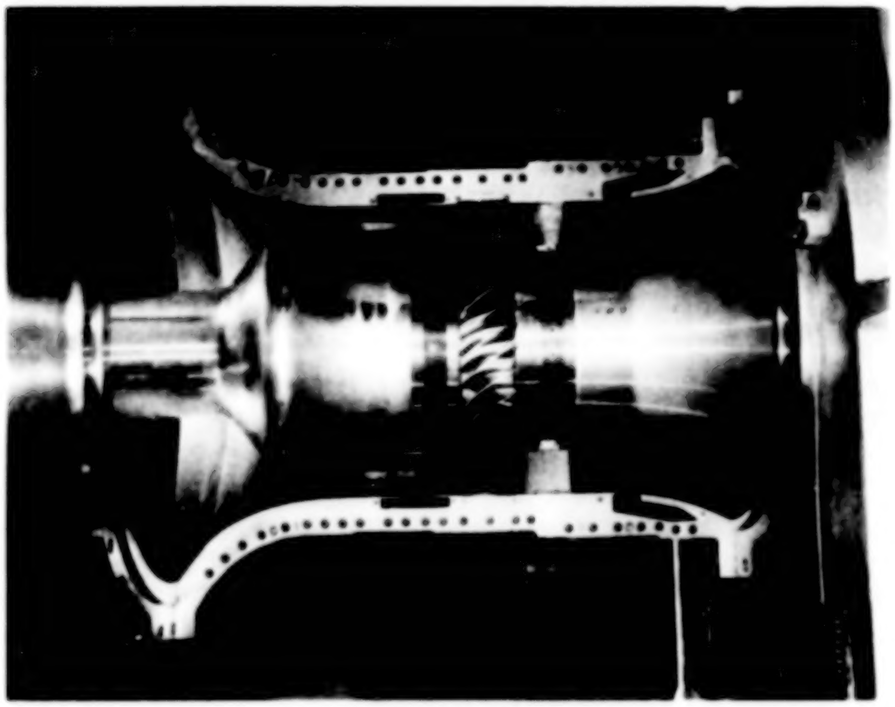
Pressure ratio	2.45
Inlet axial Mach number	2.00
Rotor tip speed	1500 ft/sec
Tip diameter	20 in.


Translating nozzle centerbody
Translating diffuser centerbody
Dry air
Bleed
Vacuum exhaust

CD-91-54403

A baseline fan stage is currently being tested at NASA Lewis to demonstrate the viability of establishing and maintaining supersonic flow through a turbomachinery stage. Detailed flowfield mapping experiments will also be conducted, the results of which will be used to validate the various computer codes used in the design and analysis process.

Supersonic Fan Test Section

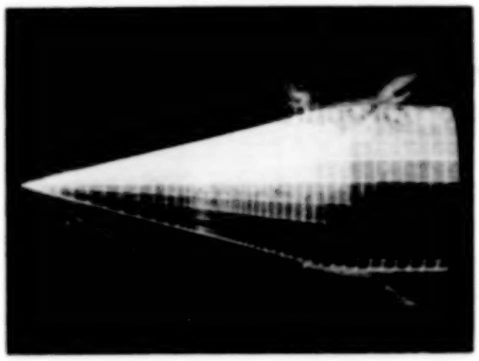


ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54404

The photograph of the supersonic fan test section shows the rotor and stator blades installed and also the hub and tip bleed regions, which can be used to vary the incoming boundary layer profiles to the fan stage.

Supersonic Through-Flow Inlet Technology



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54405

Currently underway are experimental and analytical studies of inlet concepts that would be appropriate for a supersonic fan. To the small-scale inlet tests being conducted at low- and high-speed conditions, the advanced Navier-Stokes flow solvers were applied to predict the inlet steady-state and dynamic performance characteristics. Similar combined analytical and experimental efforts will be started in FY 1991 to investigate aft inlet and fan nozzle concepts.

Concluding Remarks

- Supersonic cruise propulsion research is a growing part of NASA aeropropulsion program.
- Research efforts have both near- and far-term emphases
  - "Near term" to support IOC 2005 to 2008 HSCT
  - "Far term" to demonstrate viability of supersonic propulsion system based on supersonic through-flow fan.

CD-91-54406

Supersonic cruise propulsion research is a growing part of NASA's aeropropulsion program and is poised to provide the propulsion technologies required to maintain U.S. leadership in the international aerospace market in the 21st century.

N91-20089

OVERVIEW OF HIGH PERFORMANCE AIRCRAFT PROPULSION

Peter G. Batterton and Thomas J. Biesiadny
NASA Lewis Research Center
Cleveland, Ohio

This paper presents the basic overall scope of the Lewis High Performance Aircraft Propulsion Research and Technology effort. High performance fighter aircraft of interest include supersonic fighters with such capabilities as short take off and vertical landing (STOVL) and/or high maneuverability. The effort is primarily focused on component-level experimental and analytical research. This research is designed to provide databases for verification of design technology and for calibration of the CFD tools available for design use. Examples from each of the research areas are discussed, and a brief look at future directions for high performance aircraft research and technology is presented.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

HIGH
ANCE AIRCRAFT

This is the High Performance Aircraft Propulsion Technology Section. High performance fighter aircraft of interest include supersonic fighters with such capabilities as short take off and vertical landing (STOVL) and/or high maneuverability. This figure indicates the basic overall scope of the Lewis high performance aircraft propulsion area. The effort is primarily focused on ground-based, component-level experimental and analytical research. The research is designed to provide experimental databases for technology verification and for identification and calibration of the CFD tools available for design use. It should be noted that the advanced light-weight and high-temperature materials being developed under the joint NASA/DOD IHPTET program are key to high performance aircraft propulsion systems, and they are critical to the long-term goals and success of this program. However, because these advanced materials have many applications, they are described in the Materials and Structures Section.

High Performance Aircraft

- Overview of High Performance Aircraft Propulsion
- Highlight Topics
  - Flow visualization and hot-gas ingestion characteristics of a vectored thrust STOVL concept
  - High angle-of-attack inlets

CD 91 54194

The High Performance Aircraft Section consists of three papers: first, a general overview, which includes an overall summary of recent activities and a brief look at plans for the future; and then two papers, which highlight hot-gas ingestion and high angle-of-attack inlets. Hot-gas ingestion is a major problem for jet-lift-powered vertical landing fighter aircraft, and high angle-of-attack inlets are key to the new generation of high maneuverability fighters.

High Performance Aircraft Propulsion

- **Program Objective**

  Ready technology options for revolutionary new capabilities in future high performance aircraft

- **Program Approach**

  Validate and demonstrate enabling technologies for supersonic STOVL and high maneuverability as potential capabilities for future fighter/attack aircraft

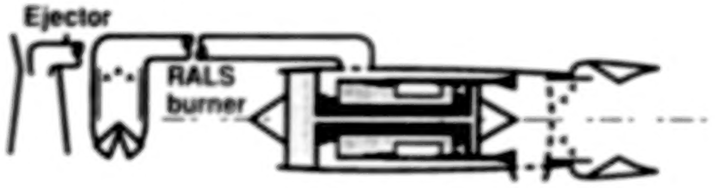
CD-91-54195

This figure states the overall objective of the High Performance Aircraft Propulsion Program, which is the development of the long-lead, high-risk, enabling technologies for these revolutionary aircraft. Specifically, the program will validate and demonstrate enabling technologies for supersonic short take-off/vertical landing (supersonic STOVL) and high maneuverability aircraft. Once validated, these technologies will become options available for Air Force and Navy consideration while the requirements for the next generation fighter are being developed. The program is being accomplished through joint studies and research with the United Kingdom (the US/UK ASTOVL Program), and with other NASA, Air Force, and Navy research organizations, including NASA Ames and NASA Langley, the Navy Propulsion Test Center, and the Air Force Wright Laboratory. However, the Air Force has recently discontinued its supersonic STOVL studies.

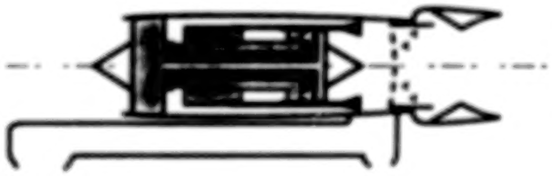
US/UK ASTOVL Studies

Propulsion Concepts
(Remote augmentors and mixed flows)

- **Fan bleed systems**



- **Exhaust bleed (MFVT/REX)**



Technology Needs:
- Diverter valve and ducts
- Hot-gas ingestion
- Integrated controls
- Nozzles
- STOVL augmentors

CD-91-[illegible]

Of major impact on our program was the US/UK ASTOVL Program joint study that considered four basic types of supersonic STOVL fighters (ref. 1). These included vectored thrust, ejector augmentor, remote augmentor, and tandem fan jet lift concepts. Although there are others, these four single-engine concepts were believed to represent the best alternatives for the studies. The study results defined the technologies that needed to be developed under the program. The figure above summarizes the specific technology needs of the propulsion system, as identified by the studies. Although the studies did not identify a specific propulsion concept for ASTOVL, they did identify the propulsion features that can lead to the best potential aircraft. The two main features are (1) remote augmentors, or lift jets, which allow greater aircraft design flexibility, and (2) mixed flow exhaust through a single rear nozzle, which permits somewhat better performance for up and away flight. The key propulsion technology needs for these systems include diverter valves and ducts, hot-gas ingestion alleviation and avoidance, integrated controls, vertical lift nozzles, and STOVL augmentors.

High Performance Aircraft Propulsion–Plan Matrix

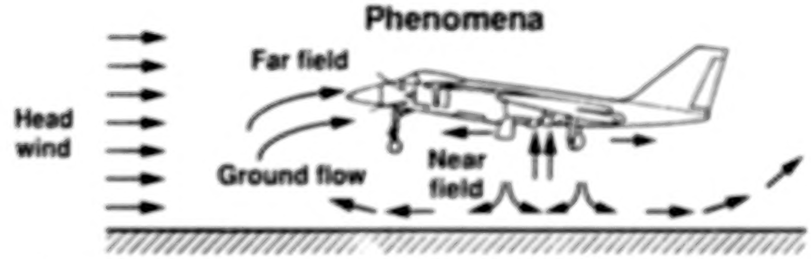
Research Area		Supersonic STOVL		High Maneuverability	
		Experiment	Analysis	Experiment	Analysis
Hot-gas ingestion		✓	✓	N/A	
Ventral nozzles		✓	✓		
Off-takes, valves, and ducts		✓	✓		
Integrated flight/ propulsion controls		✓		(STOVL effort applicable)	
Thrust augmenting ejectors		✓	✓	N/A	
Inlets	High alpha &short diffuser	(High maneuverability effort applicable)		✓	✓

CD-91-54187

Because of the outcome of the US/UK ASTOVL study, the scope of the Lewis program includes the first five research areas listed in this figure. Inlet technologies have been added because they represent an essential technology need for high angle-of-attack/high maneuverability propulsion systems. Of the five supersonic STOVL technology efforts, we believe the integrated controls methodology is generally applicable to the high maneuver area as well. Also, even though the inlet effort is focused primarily on high angle-of-attack (high alpha) and diffuser boundary layer control, supersonic STOVL inlet systems will also require these technologies. In addition, the matrix emphasizes that each technology area includes both analytical and experimental research, except for controls, which is primarily analytical. The experimental research will provide data for verification of the analyses as well as databases of related information on performance and design requirements. The remainder of this report and the two that follow describe the status and summary results for each research area.

Hot-Gas Ingestion

Phenomena



Experimental

Develop/Validate

Techniques to predict and evaluate at model and full scale

Analytical

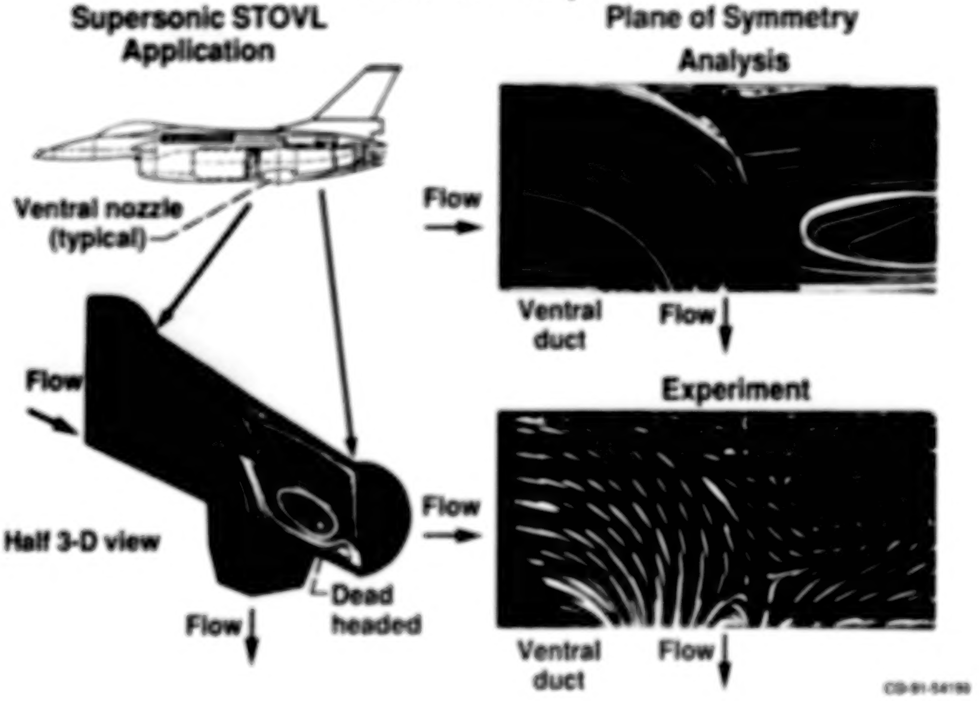
CD-81-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This figure will only introduce the subject of hot-gas ingestion. Hot-gas ingestion occurs when a lift-jet-powered aircraft approaches the ground; the jet exhaust hits the ground and either spreads out along the ground surface or is reflected back up to the underside of the aircraft. In the first case, the hot gas eventually rises or is blown back toward the aircraft where it can be ingested by the inlet and engine. In the second case, the hot gas flows around the fuselage and is ingested by the inlet. In either case this results in temperature distortion and loss of engine stability margin and/or performance. The objective of the hot-gas ingestion research at Lewis is to develop a database for use in studying the impact of hot-gas ingestion on engine performance and stability and for calibrating prediction techniques. Although configuration dependent, methods for controlling hot-gas ingestion have also been investigated. The paper that follows will present significant results of the experimental and analytical hot-gas ingestion research. The analytical effort is being accomplished by the Internal Fluid Mechanics Division. The lower right plot in the figure shows a computed ground plane temperature pattern that is representative of their work (ref. 2).

Ventral Nozzles

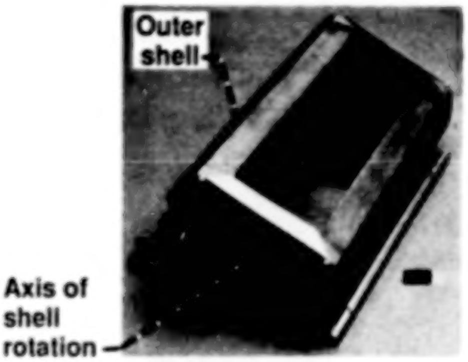
PARC-3D Analyses

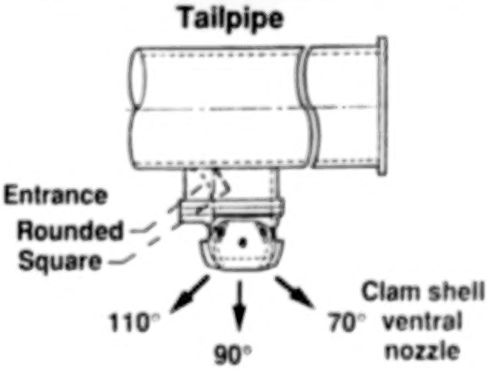


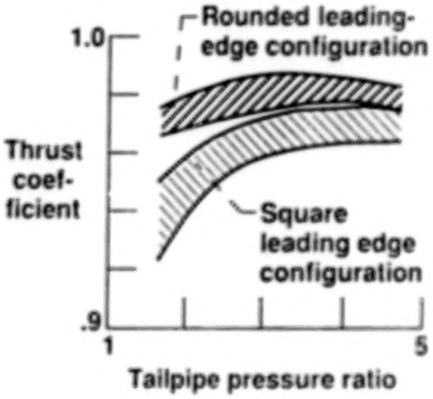
Ventral nozzles can provide vertical lift thrust behind the aircraft center of gravity during hover and low-speed operations. Their main advantage is that the thrust axis can be located closer to the center of gravity than can an aft deflecting nozzle, which allows greater aircraft design flexibility. The initial focus of the ventral nozzle research was to assess an available, full Navier-Stokes CFD code, PARC-3D, for predicting the details of the ventral nozzle internal flow field. The approach was to design a simple experiment that included the key flow physics, but was relatively easy to grid for the CFD analysis (ref. 3). In the lower left of the figure is a half section of this arrangement, with computed flow stream lines shown on the plane of symmetry. The tailpipe was modeled experimentally and analytically as a constant area duct terminated with a flat plate simulating a blocked main nozzle. The ventral nozzle was modeled as a two-dimensional convergent nozzle located at right angles to this main duct. Experimental paint streaks were obtained by using a flat plate installed on the plane of symmetry in the tailpipe duct. The right half of the figure directly compares the paint streaks and the analytical stream lines. Qualitatively, the results are very similar. Nozzle performance parameters such as mass flow, thrust, thrust angle, thrust coefficient, and flow coefficient were all predicted by PARC-3D (ref. 3) to better than 1 percent of the experimental values. The CFD analysis also identified a pair of vortices near the upstream surface and an accompanying large separation on the surface of the two-dimensional ventral nozzle duct. These disturbances caused the exit flow and thrust to be turned about 5° more than the mechanical angle of 90°. They also caused most of the approximately 5-percent pressure loss for the system.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Vectoring Ventral Nozzles







Effects of Ventral Entrance Geometry

Leading edge	Overturning, degrees
Square	5
Rounded	2

CD-91-54200

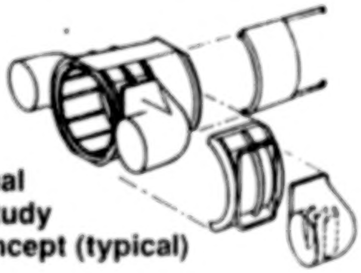
To assist transition between hover and wing-borne flight, a vectoring capability is desired in ventral nozzles. One possible method to provide thrust vectoring would be to implement a clam-shell type of two-dimensional convergent nozzle. The previously described ventral nozzle experiment was modified (ref. 4) to incorporate this technique. Whereas in a production design each of the outer shells could be independently actuated, for this experiment the exit area and the two outer shells were connected, but could be rotated ±20° from the vertical. A rounded entry to the ventral nozzle was also investigated to reduce the internal pressure losses due to the separations discussed on the previous figure. Results of this experiment are shown in the right half of the figure above. For the thrust coefficient, results are shown in bands rather than individual points because of the variation in thrust coefficient due to the vector angle. However, the main point to be made here is that the thrust coefficient can be improved with the simple aerodynamic rounding of the upstream entrance to the ventral nozzle. Also, the rounding reduces the overturning of the thrust vector from about 5° to about 2°. A CFD analysis of this configuration is not available, but we would expect the vortices and separations to be significantly reduced by this simple treatment.

Off-Takes, Ducts, and Valves

Small-Scale Water Tank Experiment

Critical Technologies Needing Data

- Pressure losses/flow path
- Cooling/materials
- Transition control
- Sealing (leakage)
- Weight

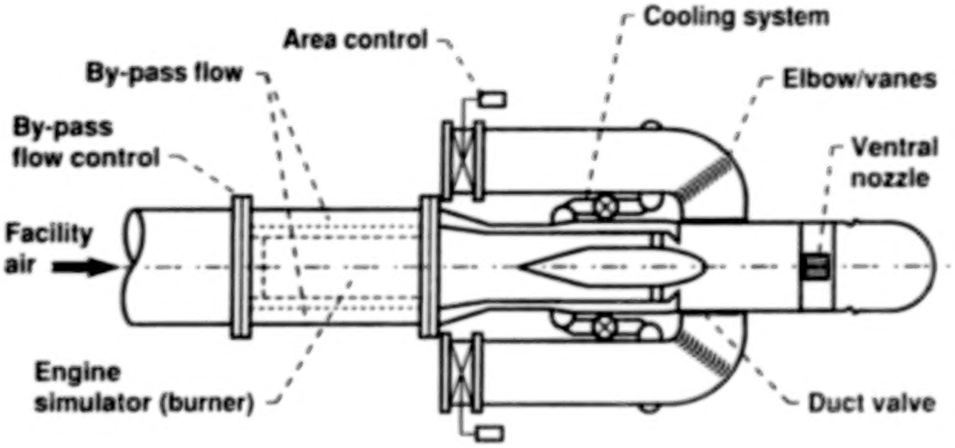


Conceptual
Design Study
Valve Concept (typical)

CD-91-54201

A relatively new research area, and key technology area for supersonic STOVL, is high performance and low loss exhaust gas off-takes and transport ducts that deliver the flow to the vertical thrusting systems. Recognizing a need for further study in this area, we contracted with Pratt & Whitney and McDonnell Aircraft to experimentally study basic flow phenomena in off-takes in a water tank. A laser light sheet was used to visualize the flow. Also, the bypass and core flows were independently controlled, as were the location of the mixing plane, flow splits, and so on. One of the configurations is shown in the upper left of the figure; in this case, mixing was not complete and the dark areas, which represent high-temperature gas, reach the outside off-take wall. Conceptual design studies of blockers and off-takes were also performed under contract by Pratt & Whitney. One of their conceptual designs is shown in the lower left of the figure. Based on the results of these two studies, the needed critical technologies and design databases were identified.

Hot-Flow Model Test Rig Concept



Variables

- Engine by-pass ratio
- Flow off-take configuration
- Cooling flows
- Duct/turning vane configuration
- Ventral nozzle size/location

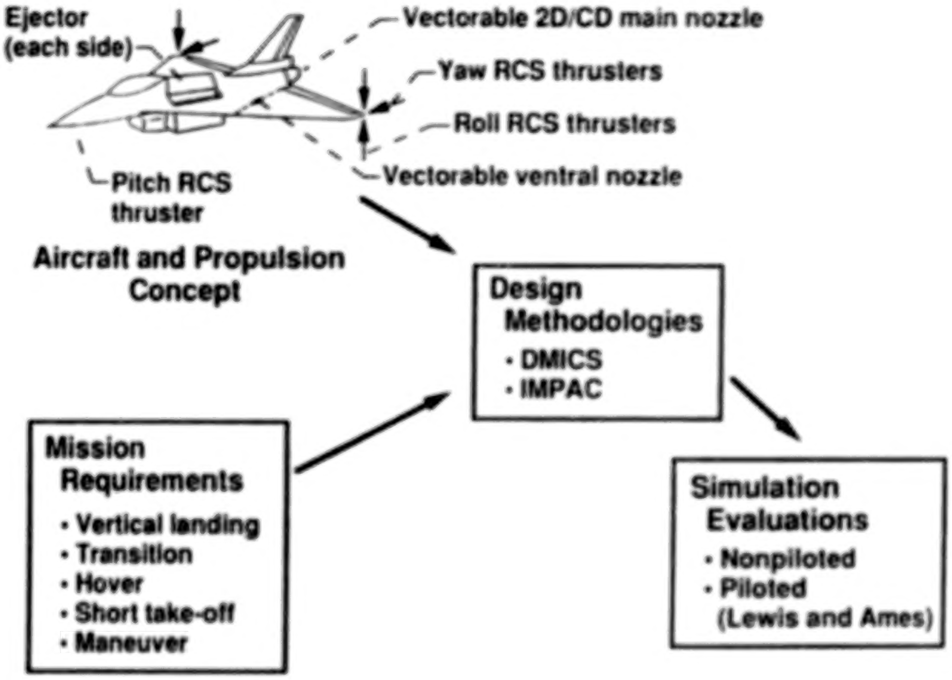
Typical Results

- Heat transfer data for cooling system design
- Flow pressure/temperature profiles
- Fan & core stream mixing
- Effect of nonsymmetric lift nozzle area variation

CD 91 54202

The results of the aforementioned conceptual design studies of valves and off-takes were expanded to define the requirements for an approximately half-scale, heated, dual-flow test rig. This rig, which will be used to develop the desired experimental database for valve and off-take designs, is shown here in conceptual form. A modification to the Lewis Powered Lift Facility is being made to incorporate a burner for simulating core engine exhaust gas and a mechanism for varying the bypass ratio (or flow split) of the heated and unheated streams. The test hardware will be of modular design so that different arrangements of off-takes and turning vanes as well as ventral nozzle locations can be investigated. Although we do not plan to use full temperatures (to save cost and lessen complexity), temperature differences will be sufficient to identify where cooling and/or high temperature materials are required. The Reynolds number will be approximately the same at the reduced temperature as that at full scale so that mixing effectiveness can be addressed. Basic performance data on pressure losses, flow coefficients, mixing, and such will be obtained. As stated before, the goal is to obtain data to establish design requirements or criteria, but flow data for calibration of analyses will also be obtained where possible.

Integrated Flight-Propulsion Controls Approach



Aircraft and Propulsion Concept

Design Methodologies
- DMICS
- IMPAC

Mission Requirements
- Vertical landing
- Transition
- Hover
- Short take-off
- Maneuver

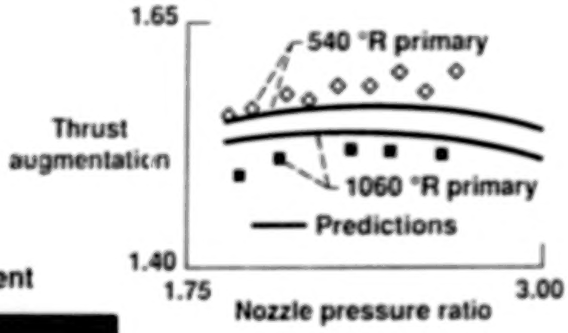
Simulation Evaluations
- Nonpiloted
- Piloted (Lewis and Ames)

CD-91-54203

The approach and status of the ongoing integrated controls effort is presented in this figure. Integrated flight-propulsion control is critical to supersonic STOVL because the propulsion system performs essentially all of the flight control functions while the aircraft is in transition and hover. Lewis is focusing on developing and demonstrating an Integrated Flight Propulsion Control system design methodology. The Air Force Design Methods for Integrated Control Systems (DMICS) (refs. 5 and 6) is the basis for our contract study by General Electric, General Dynamics, and System Control Technology. In parallel, the Instrumentation and Controls Technology Division is investigating a new optimization approach called Integrated Methodology for Propulsion and Aircraft Control (IMPAC). The Lewis IMPAC approach uses the H-Infinity control synthesis technique (ref. 7). The goal of these methodologies is a validated design procedure for integrated control which performs a systematic, top-down design based on the mission requirements and the system to be integrated. Our approach is to apply the methodologies, simulate the resulting control logic in software, and evaluate the resulting integrated control logic on nonpiloted and piloted simulations, the latter jointly with NASA Ames. The DMICS approach is currently scheduled for piloted evaluation on Ames simulators in the spring of 1992. The IMPAC approach has been evaluated in a nonpiloted decelerating transition simulation.

Thrust Augmenting Ejectors Hot Primary Results

Performance Results



Full-Scale Ejector Augmentor Component

Summary

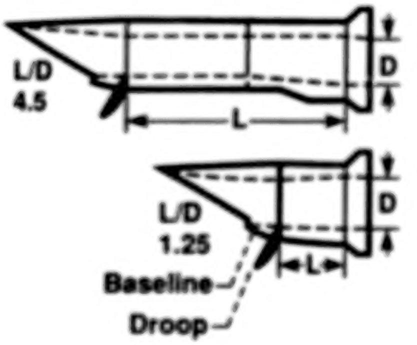
- Ejector type STOVL thrust augmentors demonstrated
- Desired performance levels achieved
- Design remains an art

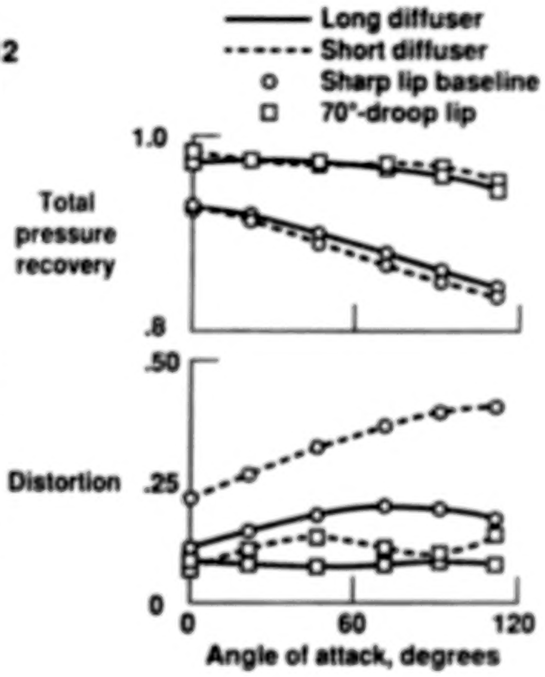
CD-91-54204

Recently, we have completed our experimental research with ejector augmentors. This research started as a joint effort with the Canadians and the de Havilland Aircraft Co. (ref. 8), but this Joint US/Canada Focused Ejector Program has been discontinued. However, Boeing/de Havilland has continued to support ejector research and has provided an ejector designed for the elevated primary gas temperatures expected in a mixed flow turbofan engine. The US/UK ASTOVL studies indicated the worth of such systems, and therefore the interest in a hot primary ejector has continued. This full-scale ejector is shown mounted in the Powered Lift Facility. Its internal physical dimensions are approximately 9 ft long by 3 ft wide at the primary nozzle exit. The curves plotted in the right of the figure summarize some of this recent testing and show predicted and measured performance at two primary nozzle gas temperatures. The predictions are based on a semiempirical code developed under contract by Sverdrup (ref. 9). Our conclusions about ejector type STOVL thrust augmentors are that, although they have been successfully demonstrated and the desired performance levels achieved, designing them remains an art. Even though we have completed our currently planned research on thrust augmenting ejector systems for supersonic STOVL, more work could be done. Specifically, ejector designs with primary nozzle pressure ratios representative of future advanced military engines need to be investigated, but this is not currently perceived as a high priority need.

Effect of Subsonic Diffuser Length
2-D Supersonic Inlet

- Free stream Mach number 0.12
- Diffuser ramp in its subsonic configuration
- Diffuser area ratio = 1.205

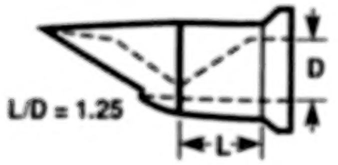


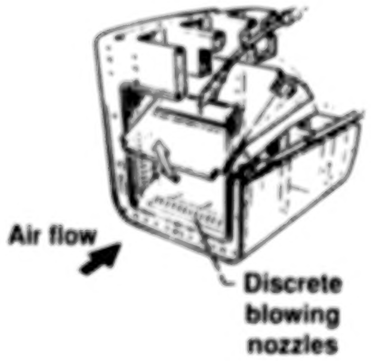


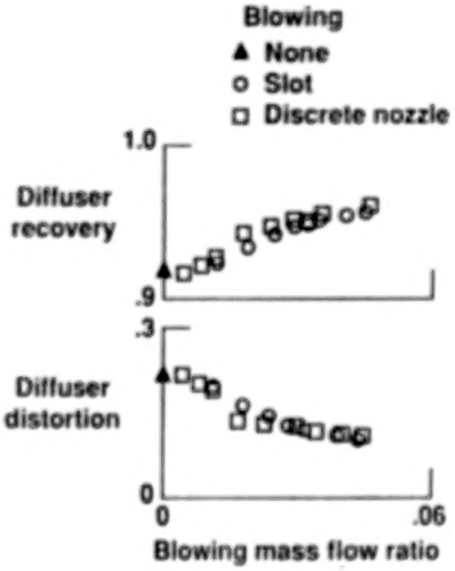
This section will cover one element of the inlet portion of the High Performance Aircraft Program - specifically, the results of the short diffuser experimental validation. An analysis performed in the early 1980's suggested that blowing could be used to avoid supersonic inlet diffuser separation. To test the concept, a previously designed and tested (ref. 10) Mach 2.2 two-dimensional inlet was modified. The inlet diffuser, which originally had a length-to-exit-diameter ratio of 4.5, was replaced by a diffuser of length-to-exit-diameter ratio of 1.25. Before the diffuser operation was assessed at simulated supersonic conditions, it was evaluated at subsonic conditions. Good quality flow must also be provided to the engine during low-speed operations and during maneuvers. Prior testing (ref. 10) investigated the angle-of-attack performance of the inlet-diffuser combination and looked at drooping the lower lip of the inlet to improve performance. This figure compares the performance of the long (baseline) and short diffuser model inlets, and a normal lip and a 70°-droop lip at free stream Mach 0.12 over an angle-of-attack range. At these conditions the pressure recovery for the short diffuser is comparable to the long diffuser. The 70°-droop lip provides considerable and comparable improvement for both inlets. In terms of distortion, however, the short diffuser does not do as well as the long diffuser, particularly for baseline sharp lower lip. The 70°-droop lip is, however, still very effective at improving distortion levels for the short diffuser inlet.

Effect of Blowing on Short Diffuser Performance
2-D Supersonic Inlet

- Diffuser ramp in supersonic configuration
- Diffuser area ratio = 1.915
- Throat Mach number = 0.77



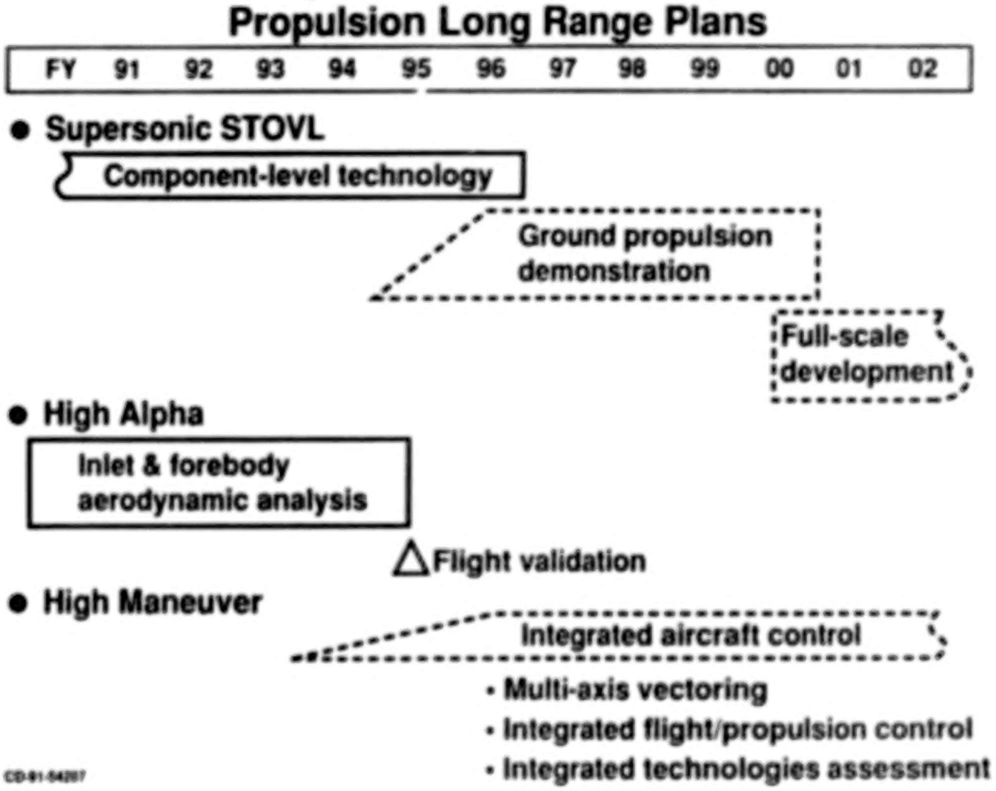




CD-91-54206

The previous figure did not address the diffuser separation issue for supersonic flight because the variable area diffuser was in its subsonic, or open, position. To simulate the effect of supersonic flow in a subsonic test, the inlet throat was closed to the nominal supersonic position, and the simulated inlet flow suction system was adjusted for a throat Mach number of 0.77; this resulted in approximately the same diffuser flow velocities as for the supersonic case. The flow in a diffuser with a length-to-exit-diameter ratio of 1.25 is guaranteed to separate from the diffuser walls. A "Panel Code" analysis (ref. 11) was used to predict diffuser separation and the amount of blowing required to avoid separation. The code predicted relatively low levels of blowing would be required to avoid such separations. In the lower left of the figure is a cut-away of the diffuser section. The discrete blowing nozzles (shown) or blowing slots were located just downstream of the inlet throat on all three fixed sides of the diffuser and the variable ramp. The plots on the right show the resultant inlet pressure recovery and distortion as a function of the blowing mass flow-to-inlet total flow ratio. The performance data for discrete nozzles and slots were essentially the same, and they approach those of the long diffuser. The experiment thus basically validated the prior analyses, and suggests blowing may offer appropriate solutions for certain types of diffuser performance problems.

High Performance Aircraft Propulsion Long Range Plans



FY 91 92 93 94 95 96 97 98 99 00 01 02

- **Supersonic STOVL**
  - Component-level technology
  - Ground propulsion demonstration
  - Full-scale development
- **High Alpha**
  - Inlet & forebody aerodynamic analysis
  - Flight validation
- **High Maneuver**
  - Integrated aircraft control
    - Multi-axis vectoring
    - Integrated flight/propulsion control
    - Integrated technologies assessment

CD-91-54207

This figure shows the overall NASA High Performance Aircraft Plan. The plan has three main elements: supersonic STOVL, high angle-of-attack (or alpha), and high maneuver technologies integration. Current plans call for continuation of the supersonic STOVL component-level technology development until an augmentation for a ground propulsion demonstration can be accomplished. We feel that such a demonstration is essential before supersonic STOVL can truly be considered available for aircraft development. This segment is shown by dashed lines in the figure because the funding and commitment to such an effort requires DOD participation in the program; this commitment is uncertain at this time. The current high-alpha forebody and inlet program continues to focus on flight and ground validation of advanced three-dimensional aerodynamics analyses and will culminate in flight experiments of the F-18 High Angle-of Attack Research Vehicle (HARV) with multi-axis thrust vectoring. The High Maneuver Integration area is proposed as a follow-on to integrate the various technologies evolving from the other programs. From the propulsion aspect, this will focus on multi-axis vectoring nozzles and integrated flight-propulsion control for the high maneuver flight regime.

Concluding Remarks

- Excellent progress on developing tools and understanding of key technology needs
- For hot gas ingestion, Lewis has developed unique capability and facility
- Internal 3-D Navier-Stokes codes are showing great promise for predicting flow inside complex geometries
- Ejector thrust augmentors provide desired levels of thrust when applied carefully
- As predicted by "Panel Codes," supersonic inlet diffuser boundary layer separation can be controlled
- Progress continues to be made; however there is still much work to be done

CD-91-[illegible]

In summary, for supersonic STOVL, excellent progress continues to be made in developing design tools and in better understanding the key technology needs. Further, as will be discussed in the following paper, NASA Lewis has a unique capability and facility for accomplishing hot-gas ingestion research. In all aspects of the program, internal three-dimensional Navier-Stokes flow codes are showing great promise for predicting aerodynamic performance inside complex geometries. Ejector thrust augmentors can provide desired levels of thrust when applied carefully. However, each application is unique. As predicted with "Panel Codes," supersonic inlet diffuser boundary layer separation can be controlled. Although we believe excellent progress is being made, there is still much work to be done.

REFERENCES

1. Levine, J.; and Inglis, M.: US/UK Advanced Short Takeoff and Vertical Landing Program (ASTOVL). AIAA Paper 89-2039, July 1989.

2. VanOverbeke, T.J.; and Holdeman, J.D.: A Numerical Study of the Hot Gas Environment Around a STOVL Aircraft in Ground Proximity. AIAA Paper 86-2882, July 1988 (Also, NASA TM-100895).

3. Smith, C.F.; and McArdle, J.G.: Analysis of Internal Flow in a Ventral Nozzle for STOVL Aircraft. AIAA Paper 90-1899, July 1990 (Also, NASA TM-103123).

4. Esker, B.S.; and McArdle, J.G.: Performance Characteristics of a One-Third-Scale, Vectorable Ventral Nozzle for SSTOVL Aircraft. AIAA Paper 90-2271, July 1990 (Also, NASA TM-103120).

5. Smith, K.L., et al.: Design Methods for Integrated Control Systems. AFWAL TR-86-2103, Wright Patterson AFB, OH, Dec. 1986.

6. Shaw, P.D., et al.: Design Methods for Integrated Controls Systems. AFWAL TR-88-2061, Wright Patterson AFB, OH, June 1988.

7. Garg, S., et al.: H-Infinity Based Integrated Flight/Propulsion Control Design for a STOVL Aircraft in Transition Flight. AIAA Paper 90-3335, Aug. 1990 (Also NASA TM-103198).

8. Corsiglia, V., et al.: Large-Scale Wind Tunnel Tests of an Ejector-Lift STOVL Aircraft Model: AIAA Paper 89-2905, July 1989.

9. Drummond, C.K.: A Control-Volume Method for Analysis of Unsteady Thrust Augmenting Ejector Flows. NASA CR-182203, 1988.

10. Mark J.L.; McGarry, M.A.; and Reagan, P.V.: Research on a Two-Dimensional Inlet for a Supersonic V/STOL Propulsion System. NASA CR-174945, 1984.

11. Hess, L.J.; and Friedman M.D.: Calculation of Compressible Flow in and About Three-Dimensional Inlets With and Without Auxiliary Inlets by a Higher-Order Panel Method. NASA CR-168009, 1982.

N91-20090

FLOW VISUALIZATION AND HOT GAS INGESTION CHARACTERISTICS OF A VECTORED THRUST STOVL CONCEPT

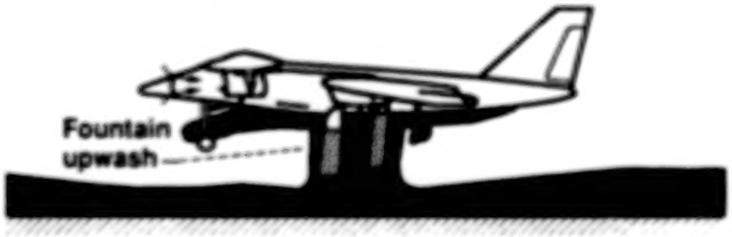
Albert L. Johns, George H. Neiner, Timothy J. Bencic,
and David M. Fricker
Lewis Research Center
Cleveland, Ohio

Jerome T. Kegelman
McDonnell Douglas Research Laboratories
St. Louis, Missouri

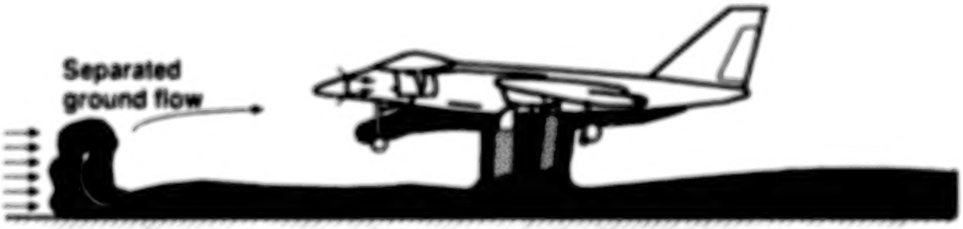
Hot gas ingestion, the entrainment of heated engine exhaust into the inlet flow field, is the key development issue for advanced short takeoff and vertical landing aircraft (refs. 1 and 2).

A 9.2-percent scale short takeoff and vertical landing (STOVL) hot gas ingestion model was designed and built by McDonnell Douglas Corporation (MCAIR) and tested in the NASA Lewis Research Center 9- by 15-Foot Low Speed Wind Tunnel (LSWT). The test was conducted over a range of headwind velocities from 10 to 23 kn and nozzle exhaust temperatures from 500 to 1000 °F. The model was also tested over a range of model heights above the ground plane (0.20 to 12 in.) and a range of nozzle pressure ratios from 1.3 to 4.00. A copper vapor laser was used to create an illuminated flow field for flow visualization with the model in and out of ground effects. This paper will present results showing the flow field visualization which occurs when the model was in and out of ground effects. The effects of hot gas ingestion on the compressor face temperature rise and several other parameters will also be presented. The environmental effects of the hot gas on the ground and its effect on the acoustic loads as a function of the model height above the ground will also be presented.

Two Main Sources of Hot Gas Ingestion
Near and Far Field



Near-field hot gas ingestion due to fountain upwash

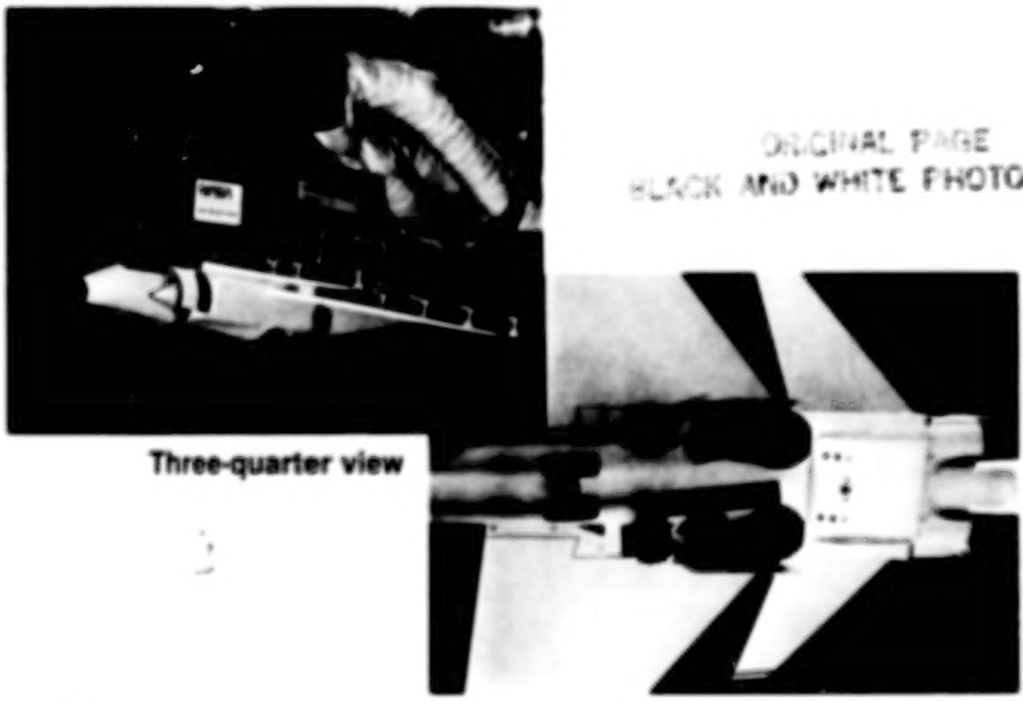


Far-field hot gas ingestion due to separated ground flow

CD-81-[illegible]

Hot gas ingestion (HGI) can be categorized as near- and far-field phenomena. The near-field hot gas problem results from the hot exhaust jet impinging on the ground plane and the jet flowing in all directions. When this jet encounters another jet, it will set up a fountain which, if under the fuselage, will interact with the fuselage and flow forward into the inlet intake region. The near-field hot gas ingestion is generally the primary source of the hot gas ingestion. The near-field hot gas ingestion is a function of the model height above the ground plane (ref. 3). The far-field hot gas ingestion occurs when the ground jet flow separates from the ground ahead of the model and gets blown back into the inlet flow field. The far-field hot gas ingestion is a function of the headwind. The magnitude of the far-field hot gas ingestion is greatly reduced in comparison to that of the near field.

Model 279-3C



Three-quarter view

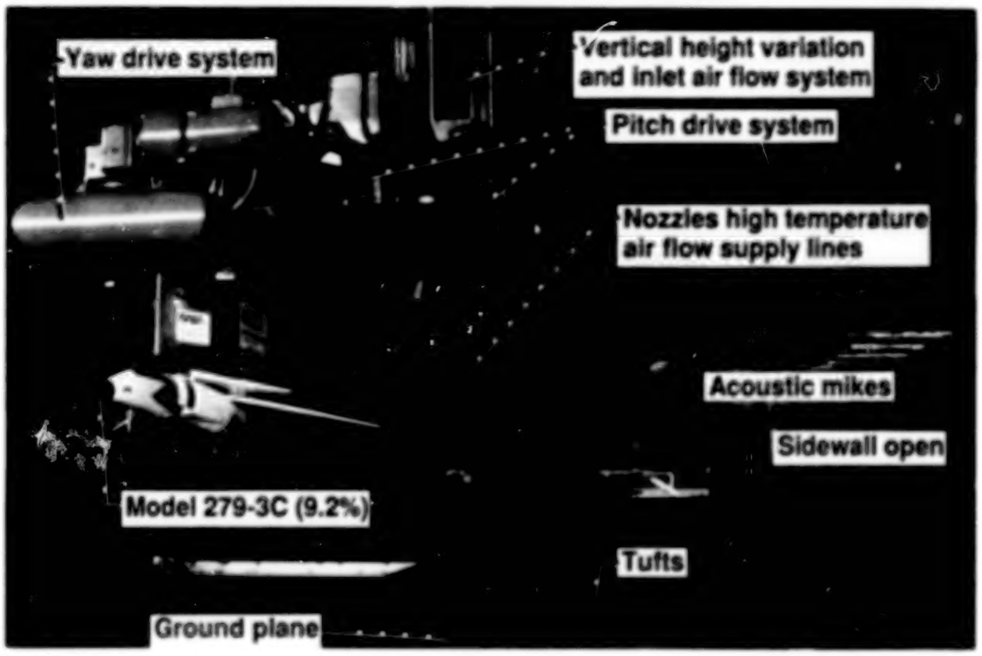
Undersurface view

CD-91-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The undersurface of the hot gas ingestion (HGI) model, 279.3C, and a three-quarter view are shown. The HGI model consisted of five major subassemblies: the forward fuselage, center fuselage, aft fuselage, wings, and canards. The forward fuselage contained the main inlet and a translating cowl auxiliary inlet, which makes up the bifurcated inlet system. The inlet suction duct is part of the suction system which was used to create inlet (compressor face) flow. The center fuselage contained the nozzle system, and high pressure hot air lines supplied the hot air to the model's four nozzles. Lift improvement devices (LIDs) were also attached to the center fuselage. The LID's consist of longitudinal strakes, sidewalls, forward fence and aft fence (optional). The LID's generally enclosed the forward and aft pair nozzles.

9- by 15- Foot Low Speed Wind Tunnel Installation of Model 279-3C and "MISS"


Yaw drive system
Vertical height variation and inlet air flow system
Pitch drive system
Nozzles high temperature air flow supply lines
Acoustic mikes
Sidewall open
Model 279-3C (9.2%)
Tufts
Ground plane

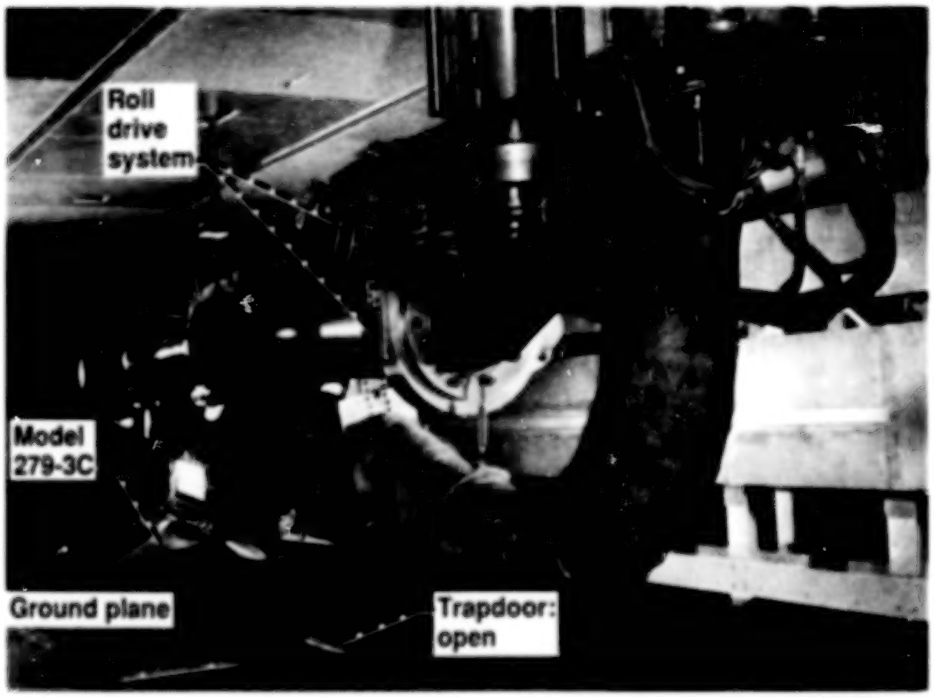
CD-91-54410

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The installation of a 9.2-percent scaled model and the supporting system in the 9- by 15-Foot Low Speed Wind Tunnel (LSWT) are shown in this figure. The NASA Lewis 9- by 15-foot LSWT was used to develop the hot gas ingestion data base. The 9- by 15-foot LSWT (constructed within the return leg of the 8- by 6-Foot Supersonic Wind Tunnel (SWT)), is 9 ft high, 15 ft wide, and has a 25-ft-long test section. The tunnel velocities for hot gas testing ranged from 8 to 23 kn.

The model supporting system includes a remotely controlled model integrated support system (MISS) that has four degrees of freedom (model height, yaw, pitch, and roll). The ground plane contained pressure, temperature measurements, and tufts for flow visualization. The sidewalls were open to vent lateral flow out of the test section, thus preventing flow circulation. Acoustic mikes and transducers (canard and wing) were used for noise and structural acoustic measurements, respectively.

Aft View of Model 279-3C With "MISS" Installed in 9- by 15- Foot LSWT

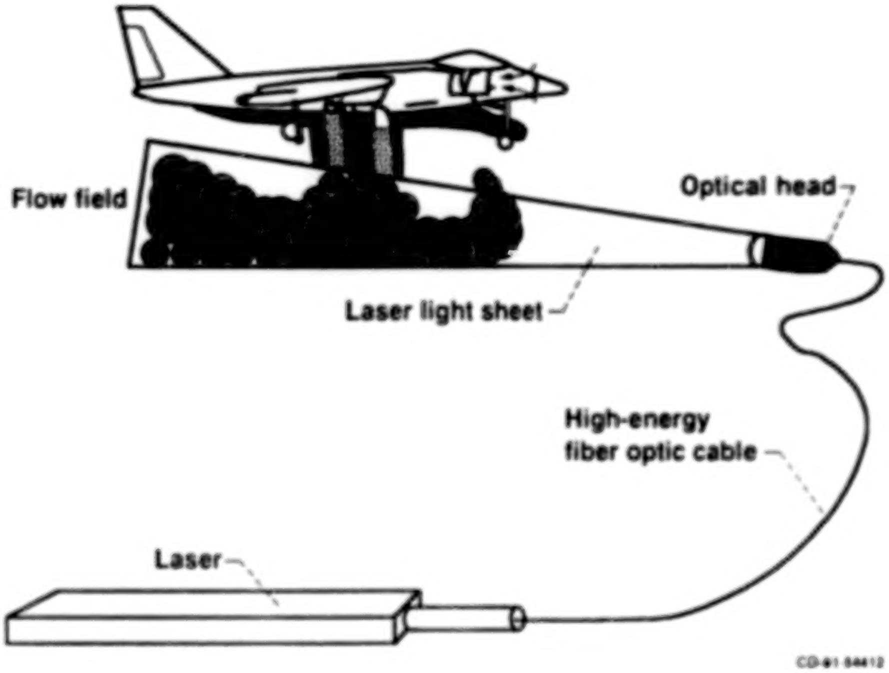


CD-91-54411

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

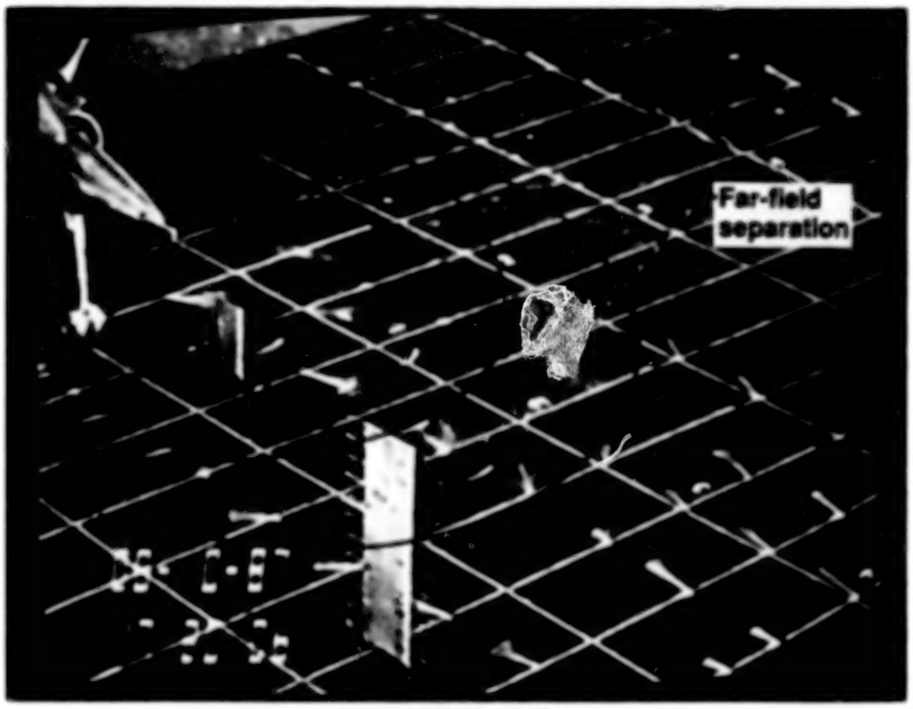
The aft view shows the high-temperature air flow supply lines of the nozzles and the aft view of the MISS roll drive system. The pitch, roll, yaw, and height variation systems formed the inlet suction path. A sliding trap door (with a scavenging system) was located in the ground plane under the region of the model nozzles. The trap door was open when setting nozzle conditions to avoid artificially heating the model undersurface and ground environment. The model nozzle hot air flow was pulled through the trap door opening, down the duct located under the ground plane, and exhausted downstream of the test section by a pair of ejectors. When a data point was taken, the trap door was closed and the ejectors were automatically shut off.

Laser Sheet Illumination System


Flow field
Optical head
Laser light sheet
High-energy fiber optic cable
Laser
CD-91-54412

A copper metal vapor laser was used to generate the illumination system for the flow field visualization. Atomized water was used as the seeding agent. The light from the laser was transmitted to the test section by means of a high-energy fiber optic cable. A traversing rig was used to position the optical head (sheet) at various stations in any of the three planes. The laser operates in the 510.6- and 578.2-nm wavelength. This was the first use of a laser sheet system in NASA Lewis Research Center propulsion wind tunnels.

Tufts Showing Far-Field Wall Jet Separation

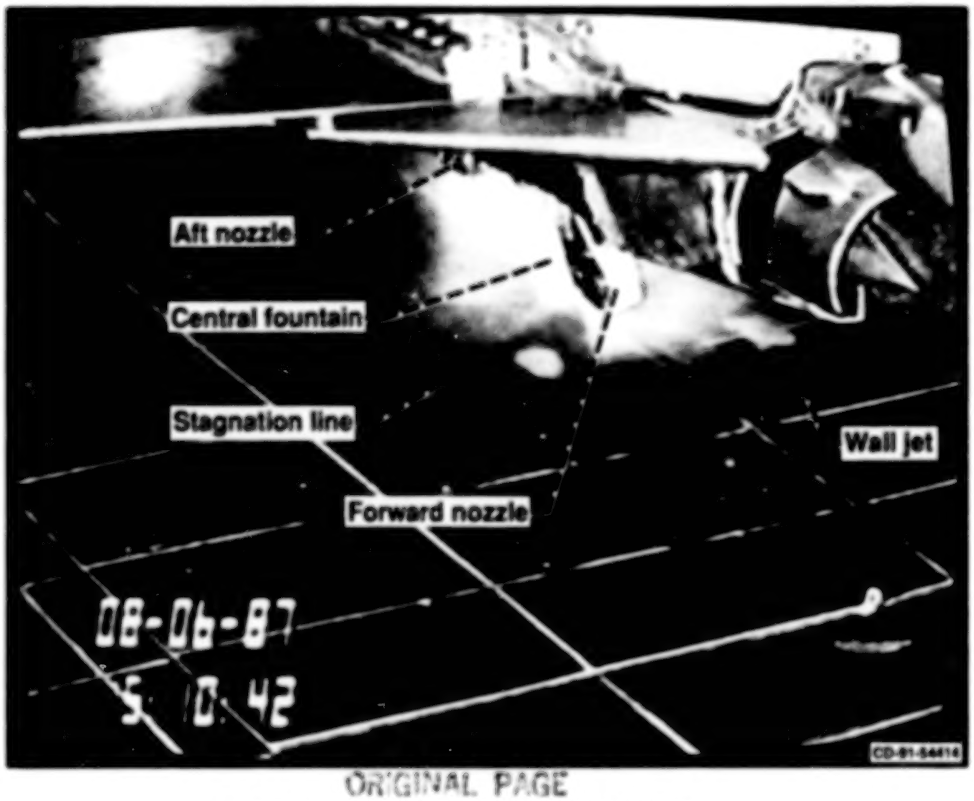


ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54413

Tufts were used on the ground plane to determine the mid- and far-field ground flow characteristics. The tufts pointing toward the model are under the influence of the freestream headwind velocity. The tufts pointing away from the model are under the influence of the nozzle wall jet flow. At the region where they meet and the nozzle wall jet flow is slightly less than the headwind velocity, the wall jet separates and the tufts are unsteady (pointing away from and toward the model as a function of time). The region of nozzle wall jet influence is bounded in the lateral position by headwind velocity. The far-field separation can be observed where the tufts are unsteady.

White Light and Water Mist
Showing Near-Field Flow Characteristics

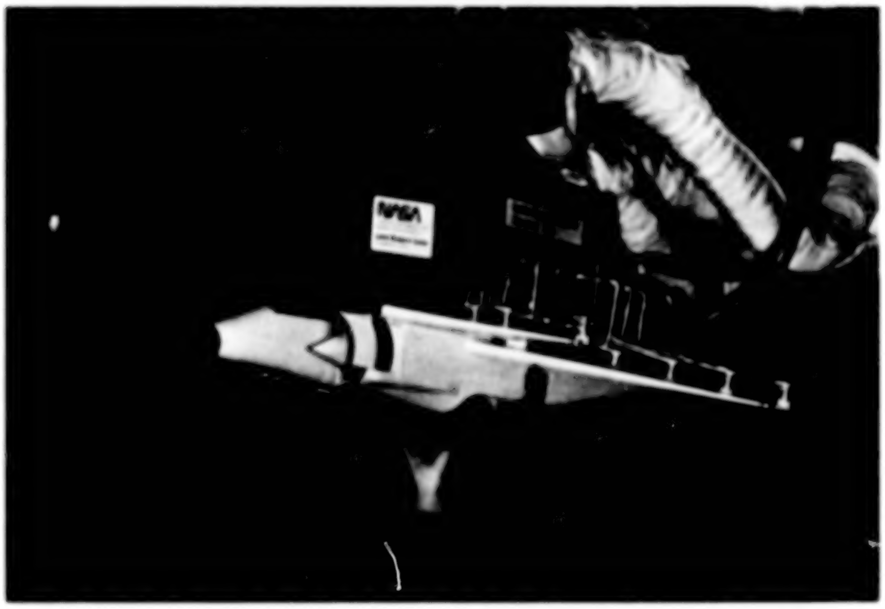

Aft nozzle
Central fountain
Stagnation line
Wall jet
Forward nozzle
08-06-87
5 10 42
CD-91-54414

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Flow visualization using white light and water at high pressure (atomized) produced a general (gross) view of the near-field flow characteristics as shown in this figure. The wall jet, stagnation line, and central fountain can be observed, although the details of the flow field are lost.

Laser Sheet in Spanwise Direction

Near-Field Flow Characteristics

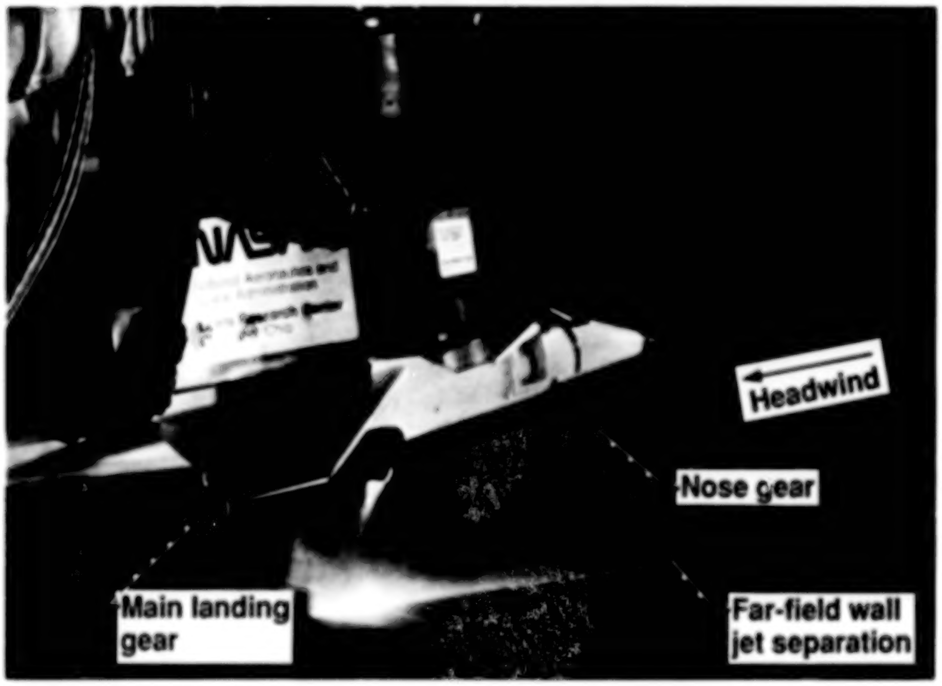


CD-91-54415

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

To obtain detailed information of the near- and far-field flows, it is necessary to use a laser sheet as the illumination source. In this figure, the laser sheet system is located in the spanwise direction. This position shows details of the near field and the laterally flowing wall jet. The laser sheet can be positioned over a range of axial stations to show the flow details, including the shocks in the nozzle jet plume.

Laser Sheet in Streamwise Direction Far- and Near-Field Flow Characteristics

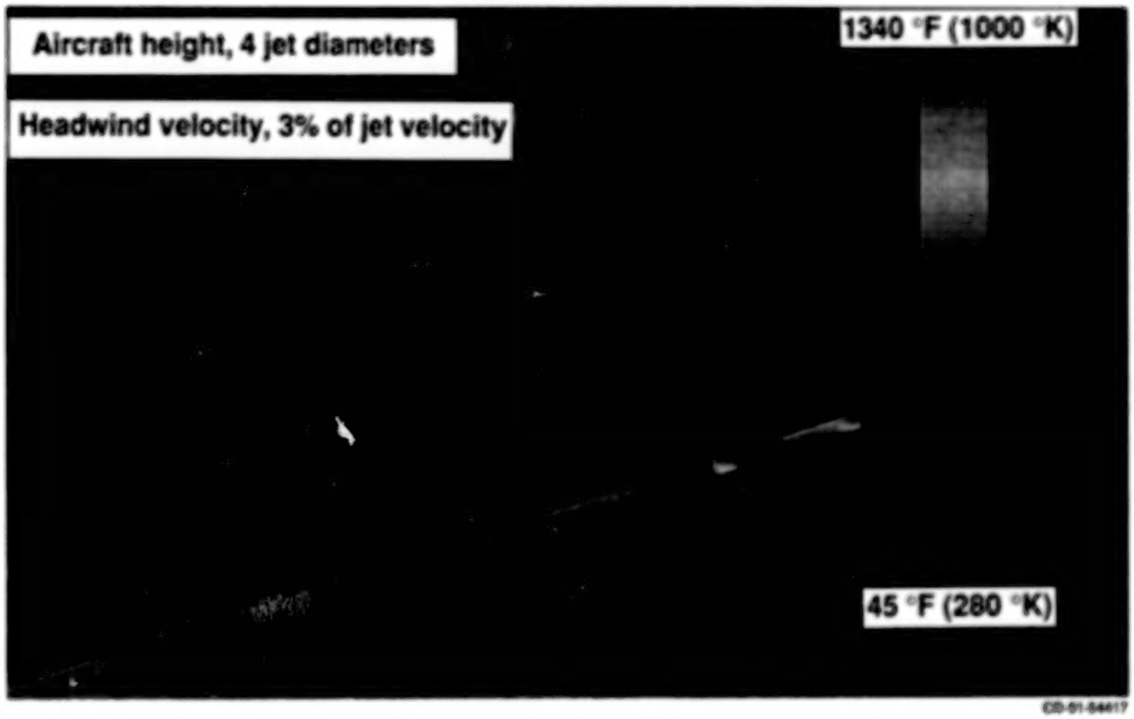

Headwind
Nose gear
Main landing gear
Far-field wall jet separation

CD-91-54418

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The laser sheet position in the streamwise direction will give a view in both the far and near fields. As a result, the effect of the height above the ground plane and the far-field interaction can be observed. The fountain between the forward and aft nozzles can also be seen.

CFD Particle Traces Showing Ingestion


Aircraft height, 4 jet diameters
Headwind velocity, 3% of jet velocity
1340 °F (1000 °K)
45 °F (280 °K)

CD-91-54417

ORIGINAL PAGE IS
OF POOR QUALITY

Using a state-of-the-art computer code, the flow field around a simple model of a short takeoff and vertical landing (STOVL) aircraft can be calculated. The computational fluid dynamics (CFD) code solves the turbulent, Reynolds-averaged, Navier-Stokes equations in three dimensions, using a multigrid technique. The CFD codes uses a k-ϵ turbulence model. Shown is a particle tracing of the jet emanating from the nozzles.

Image Enhancement

Video gray level output

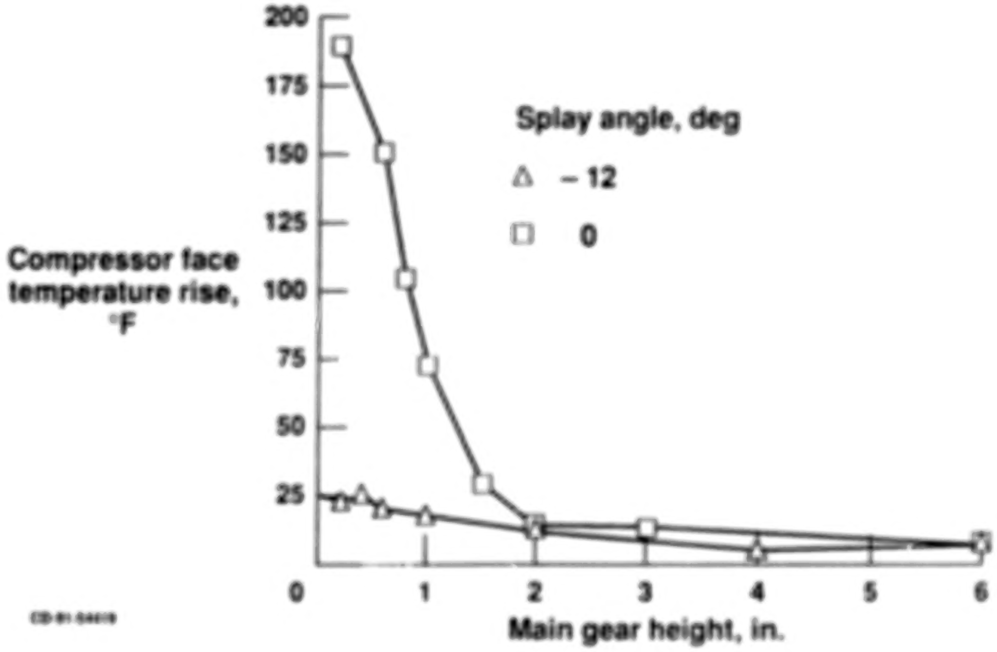
Color enhancement

CD-81-[illegible]

During the flow visualization test, a computer image enhancement program was used to color code the video gray level output. This allows the recording of the dynamic effect of the upwash fountain in the near field. Also, the density of the seeding material can be correlated with the hot region of the flow field. The dark region (in the color enhancement) located in the lighter region under the model represents the hottest (most dense) air flow region.

Effect of Forward Nozzle Splay Angle on HGI

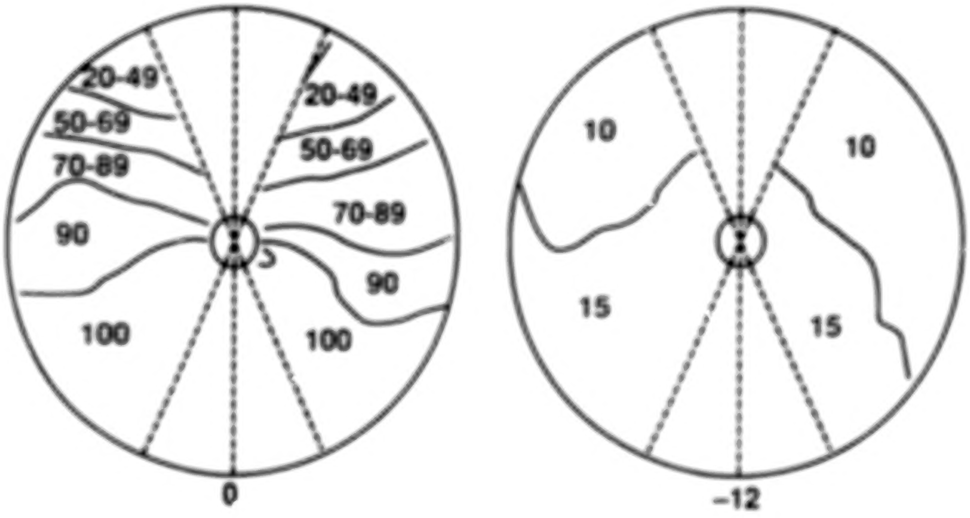
(Nozzle design pressure ratio, 3.12; headwind velocity, 10 kn; nozzle jet temperature, 900 °F)



The effect of main landing gear height above the ground plane on the compressor face temperature rise is shown for splay configurations of -12 and 0°. The negative angle indicates that the nozzles were pointed inboard. The inboard splay is an attempt to create a single jet impinging on the ground, thereby reducing or eliminating the multijet impingement effect and, hence, the upwash fountain. The results show that inboard splay tends to reduce the hot gas ingestion at the compressor face.

Contour Maps of Compressor Face Temperature Rise

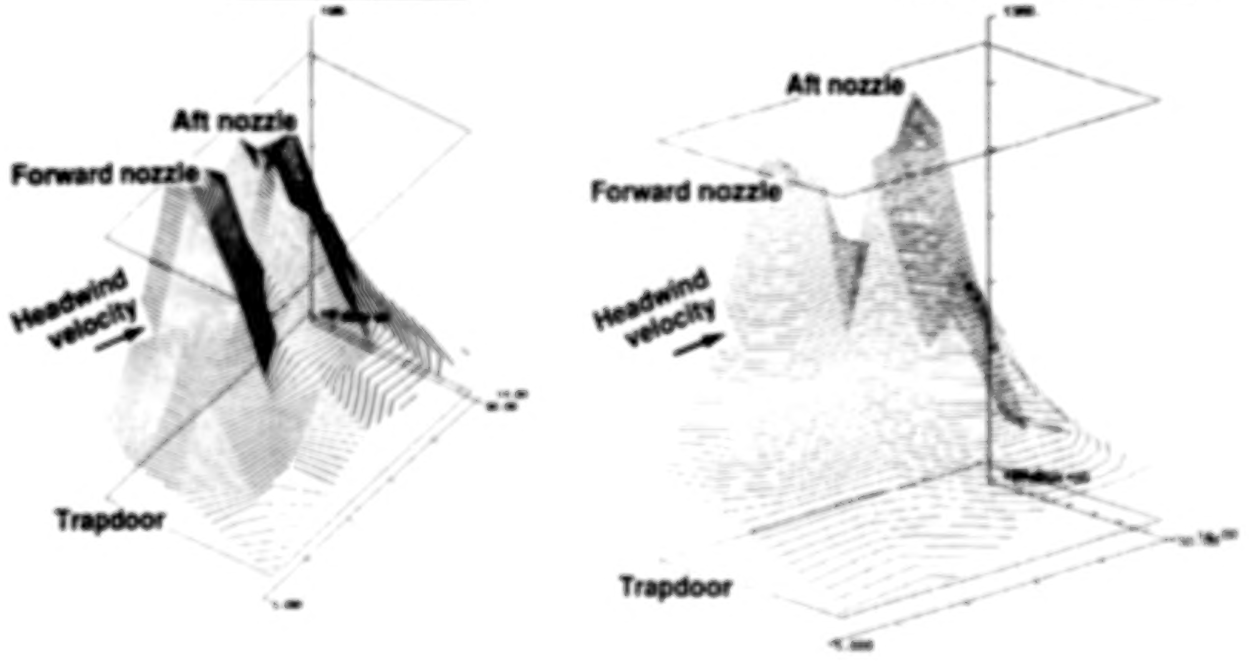
(Nozzle design pressure ratio, 3.12; headwind velocity 10 kn; main landing gear height, 0.10 in.)



Forward nozzle splay configurations, deg

Contour maps of the compressor face temperature rise at the design landing condition for the -12 and 0° splay configurations are presented in this figure. In general, the hot gas was ingested in the lower inlet section, which results in a large compressor face temperature rise for the 0° splay configuration. Splaying the forward nozzles (-12°) to create a single jet impingement significantly reduced the compressor face temperature rise.

Trapdoor Temperature [illegible]



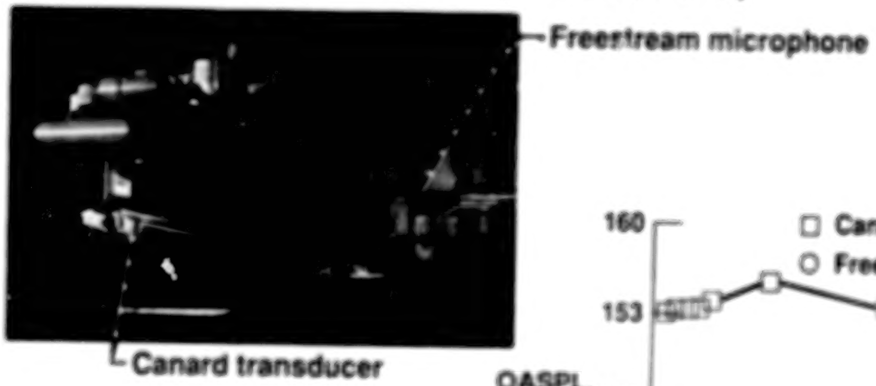
0 ° Forward Splay

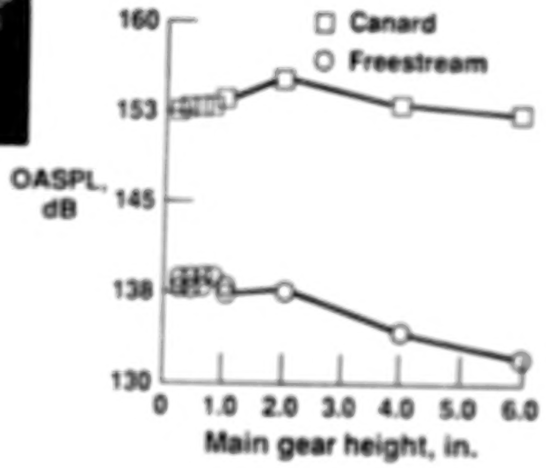
-12 ° Forward Splay

Shown here is the effect of forward nozzle splay angle on the ground plane temperature distribution. In general, the nozzles impose high-temperature footprints in the region of the nozzles. This temperature profile tends to decay rapidly as the flow moves upstream and laterally.

Model Height Effect on Acoustic Loads for 0 ° Forward Nozzle Splay

(Nozzle design pressure ratio, 3.12; headwind velocity, 10 kn; nozzle jet temperature, 500 °F)





CD-91-04422

The effect of main landing gear height on the model acoustic and near-field environment is shown for the 0° forward nozzlel splay conafaiguration. The data are presented at the design landing nozzle pressure ratio of 3.12 for the aft nozzles and 3.03 for the forward nozzles; a nozzle exhaust temperatuare of 500 °F; and a headwind velaocity of 10 kn. In general, the overall sound pressure level (OASPL) peaked when the model was approximately 2.00 in. (~20.00 in. full scale) above the ground plane. However, the near-field OASPL continued to increase until model touchdown.

FUTURE PLAN

The next hot gas ingestion testing will use the following: a thermovision system that will provide detailed data of the thermofootprint on the ground plane and data of the model surface temperature distribution; a laser Doppler velocimter (LDV) for detailed flow field measurements, which will improve CFD codes. Several advanced STOVL concepts will also be evaluated for hot gas ingestion, flow visualization, and noise characteristics.

CONCLUDING REMARKS

Data bases for both flow visualization and hot gas ingestion (HGI) have been established for the thrust vectoring short takeoff and vertical landing (STOVL) concept of model 279-3C. The use of the high-energy fiber optic laser sheet illumination system provided significant insight into the near- and far-field flow characteristics with the model in and out of ground effects. Splaying the forward nozzles inboard resulted in a significant reduction in compressor face temperature rise. Both the local freestream environment and the structural acoustic noise increase with decreasing model height above the ground plane. The model integrated support system (MISS) and the many data acquisition systems established NASA Lewis as an excellent facility for testing STOVL HGI.

REFERENCES

1. Johns, A.L.; Neiner, G.H.; Bencic, T.J.; Flood, J.D.; Amuedo, K,C.; Strock, T.W.; and Williams, B.R.: Hot Gas Ingestion Characteristics and Flow Visualization of a Vectored Thrust STOVL Concept. NASA TM-103212, 1990.

2. Johns, A.L.; Flood, J.D., Strock, T.W.; and Amuedo, K.C.: Hot Gas Ingestion Testing of an Advanced STOVL Concept in the NASA Lewis 9- by 15-Foot Low Speed Wind Tunnel With Flow Visualization. AIAA-88-3025, NASA TM-100952, 1988.

3. Amuedo, K.C.; Williams, B.R.; Flood, J.D.; and Johns, A.L.: STOVL Hot Gas Ingestion Control Technology. ASME 89-FT-323, June 1989.

N91-20091

HIGH ALPHA INLETS

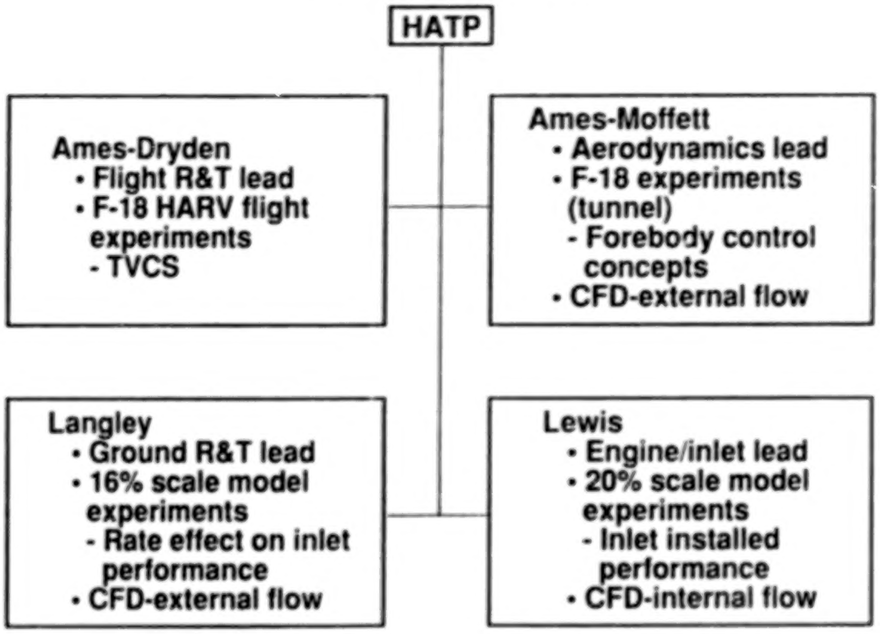
Richard R. Burley and Bernhard H. Anderson
NASA Lewis Research Center
Cleveland, Ohio

C. Frederic Smith and Gary J. Harloff
Sverdrup Technology, Inc.
Lewis Research Center Group
Cleveland, Ohio

The high alpha inlet research effort at Lewis is part of the High Alpha Technology Program (HATP) within NASA. A key goal of HATP is to develop concepts that provide a high level of control and maneuverability for high performance aircraft at low subsonic speeds and angles-of-attack above 60°. The approach, which consists of both experimental and computational efforts, utilizes the F-18 High Alpha Research Vehicle (HARV) as well as subscale models to obtain the experimental data base needed for validation of the computational codes.

As the propulsion center within NASA, the overall objective of the Lewis effort is to develop and enhance inlet technology that will insure high performance and stability of the propulsion system during aircraft maneuvers at low speeds and high angles-of-attack. This paper presents an overview of the existing Lewis technology for achieving high inlet performance at low subsonic speed/high angle-of-attack conditions and the plans to extend this technology to advanced, highly maneuverable aircraft. The discussion is divided into six parts: (1) scope of the HATP effort, (2) the inlet challenge for highly maneuverable aircraft, (3) the Lewis data base, (4) the Lewis computational effort, (5) future plans, and (6) concluding remarks.

Scope of NASA High Alpha Technology Program (HATP)



Ames-Dryden
- Flight R&T lead
- F-18 HARV flight experiments
  - TVCS

Ames-Moffett
- Aerodynamics lead
- F-18 experiments (tunnel)
  - Forebody control concepts
- CFD-external flow

Langley
- Ground R&T lead
- 16% scale model experiments
  - Rate effect on inlet performance
- CFD-external flow

Lewis
- Engine/inlet lead
- 20% scale model experiments
  - Inlet installed performance
- CFD-internal flow

CD-91-54034

The NASA High Alpha Technology Program is a cooperative effort involving Ames-Dryden, Ames-Moffett, Langley, and Lewis. Ames-Dryden is the lead center for flight research and technology. All experiments involving the F-18 HARV aircraft, including thrust vectoring control concept demonstration, will be conducted there.

Ames-Moffett is the lead center for aerodynamics. External flow experiments on a former Blue Angle aircraft without engines will be conducted in the 80- by 120-ft wind tunnel to develop forebody control concepts. Computer codes for predicting the external flow field of the F-18 HARV aircraft will be applied and evaluated.

Langley is the lead center for ground-based research and technology. Experiments will be conducted on a 16-percent scale model of the F-18 that will be pitched and yawed at high rates in the 14- by 22-ft wind tunnel to investigate the rate effect on inlet flow field and inlet performance. Computer codes for predicting the external flow field (steady state) of the F-18 HARV also will be applied and evaluated. These codes are different from those used by Ames-Moffett.

Lewis, as the propulsion center within NASA, is the lead center for the engine/inlet investigation. Experiments will be conducted in the 9- by 15-ft wind tunnel on an approximately 20-percent scale model of the HARV forebody/inlet to determine inlet-installed performance. Computer codes to predict inlet-installed performance will be applied and evaluated.

Inlet Challenge for Highly Maneuverable Aircraft

Challenge
- **Reduce inlet distortion so engine does not stall**
- **Maintain adequate thrust for maneuverability**

Types of inlet distortion for high alpha
- **Total pressure**
- **Swirl**

Vortex ingestion
Cowl lip separation
Diffuser
  – Short/straight
  – S-shape
Yaw rate at high alpha

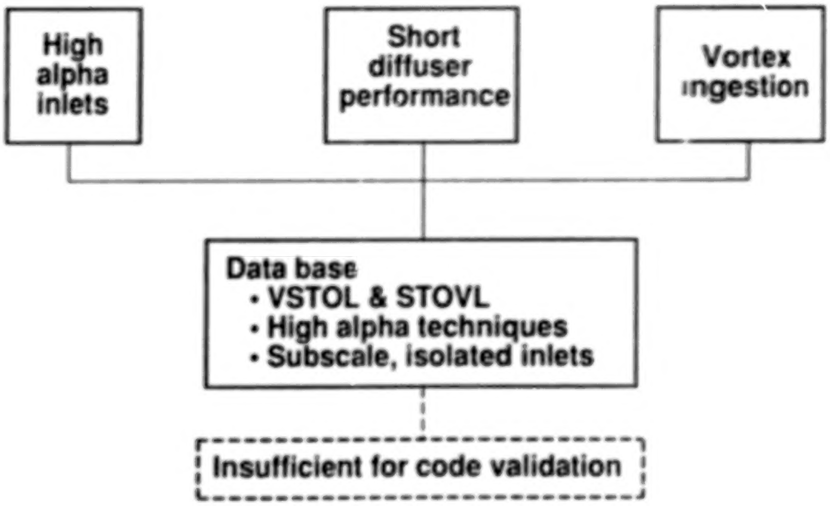
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54027

The challenge for inlets that achieve high angles-of-attack is to design them to ensure efficient delivery of airflow to the engine during maneuvers. This ensures that the engine does not stall. It also ensures that an adequate thrust level is maintained so that, in postwing stall conditions, the aircraft is still fully maneuverable. Several different types of distortion must be considered in the design of these inlets. One type, total pressure distortion, can result when the inlet is at high angle-of-attack and/or yaw conditions. It can also be encountered when an inlet is attached to a short diffuser. Another type of distortion is caused by swirl, occurring when an inlet ingests a vortex shed from some part of the aircraft or inlet lip during maneuvers. Swirl can also be encountered when an inlet is attached to an S-shaped duct and is due to secondary flow generation in the duct. Steady-state distortion of the types just discussed will be presented in this paper.

Quasi-steady-state total pressure distortion can result from a yaw rate maneuver at high angles-of-attack. More will be said about this later. Instantaneous or dynamic total pressure distortion also must be considered but will not be presented in this paper.

Data Base



High alpha inlets

Short diffuser performance

Vortex ingestion

Data base
- VSTOL & STOVL
- High alpha techniques
- Subscale, isolated inlets

Insufficient for code validation

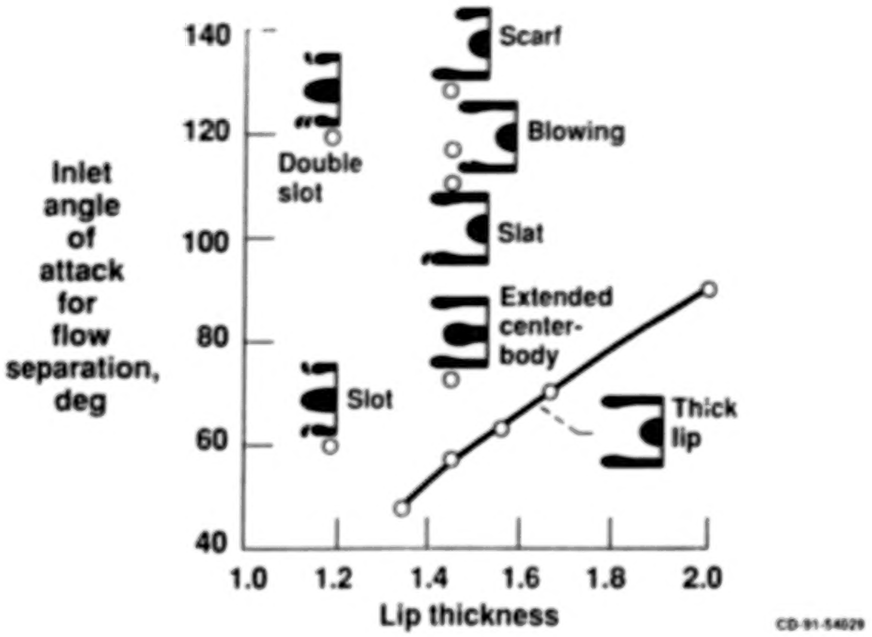
CD-91-54028

The existing data base includes past experiments to increase inlet angle-of-attack capability, improve the performance of very short, straight diffusers, and investigate the effect of ingesting vortices on engine stall. The effect of vortex ingestion is reported in reference 1. Short, straight diffuser performance was discussed in a previous paper by Peter Batterton. A sample of the data base to increase inlet angle-of-attack capability will be discussed in this paper.

The data base has been generated over the past 20 years primarily in support of subsonic and supersonic VSTOL and STOVL aircraft at low subsonic speeds and high angle-of-attack conditions. Techniques have been defined that have the potential for achieving high inlet performance at these conditions. However, the data base contains results only from subscale, isolated inlet configurations. Thus, the data base is not yet sufficient for code validation.

Data Base (continued)
High Alpha Subsonic Inlets

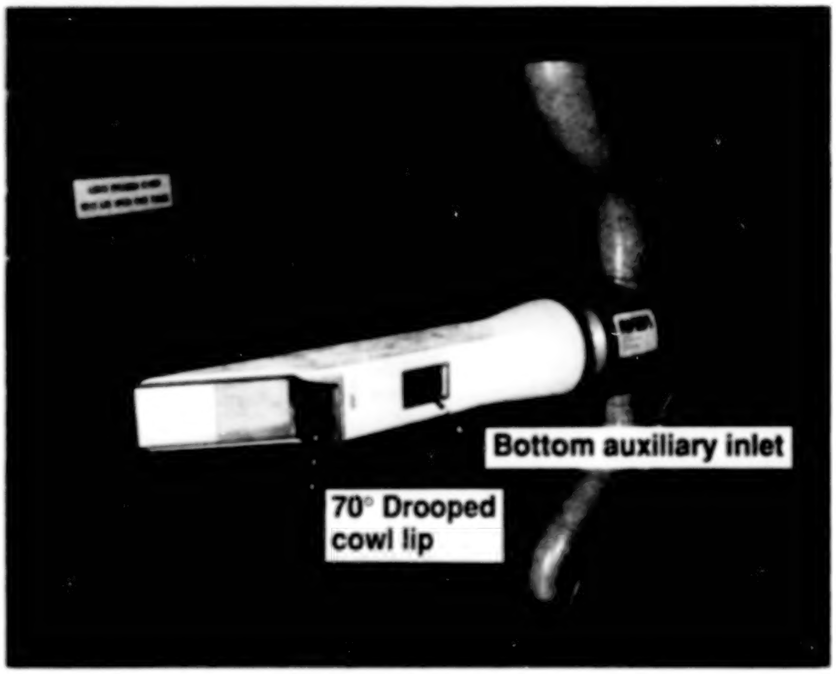
Free-stream Mach number = 0.12



A wide variety of isolated, mostly axisymmetric, subsonic inlet concepts have been tested to high angles-of-attack at low free-stream Mach numbers, as shown in this figure taken from reference 2. Desirable inlet characteristics are thin lips, to achieve both light weight inlets and good cruise performance, and high separation angles, to help insure that the aircraft maneuvers do not result in engine stall.

Techniques for achieving high angle-of-attack, shown in the figure, include increasing the lip thickness, extending the centerbody, extending the lower (windward) lip, employing active boundary layer control by tangential blowing near the inlet throat, and employing three passive boundary layer control concepts. Although increasing the lip thickness increases angle-of-attack capability, this is moving away from the goal of thin lips. For moderately thin lips, scarfing the inlet by extending the lower lip is an attractive technique. The inlet with the thinnest lip and the highest angle-of-attack capability tested employed a passive boundary layer control technique that combined a slat with a slotted inlet and was called the double slot inlet.

Data Base (continued)
2–D Supersonic Inlet in 9– by 15–ft Tunnel



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

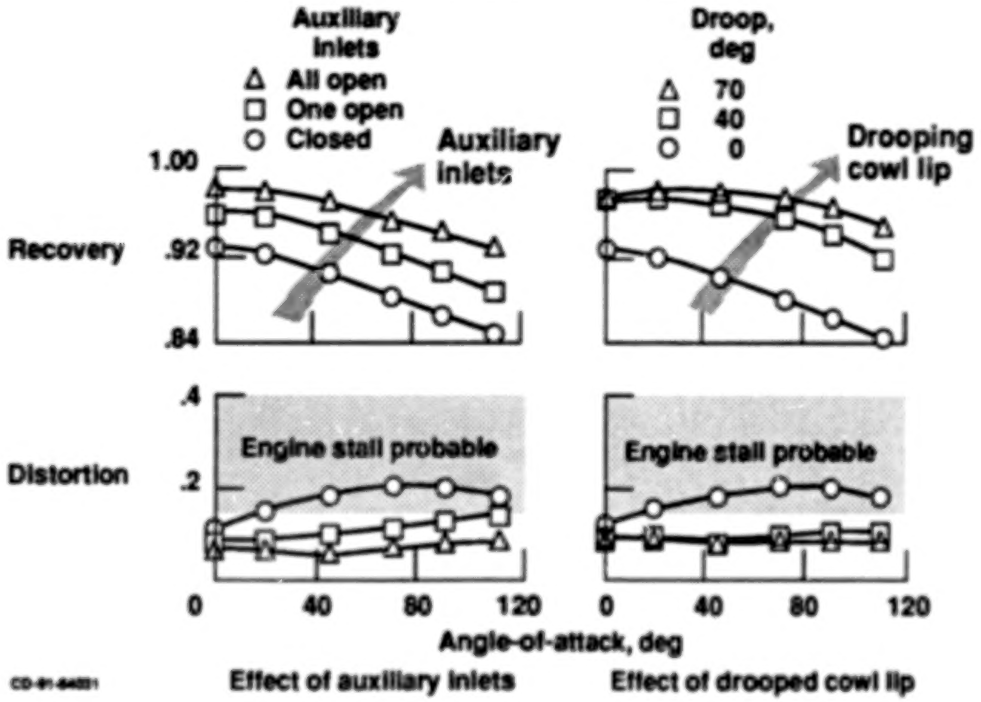
CD-81-54030

In addition to subsonic inlets, a supersonic inlet has been tested subsonically over a large range of angles-of-attack. The experimental model is shown in this figure installed in the 9- by 15-ft low speed wind tunnel. The model is a two-dimensional two-ramp inlet designed for a Mach number of 2.2.

Two different techniques were investigated for improving inlet performance at low speed/high angle-of-attack conditions: (1) drooping the cowl lip and (2) opening the auxiliary inlets located on each of the four sides of the main inlet. For illustrative purposes, the figure shows the cowl lip drooped to 70° and the bottom auxiliary inlet open. The tests, however, were conducted with either the cowl lip drooped or the auxiliary inlets open, but not in combination with each other. More details of the inlet model are given in reference 3.

Data Base (concluded)
High Alpha 2–D Supersonic Inlet

Free-stream Mach number = 0.12



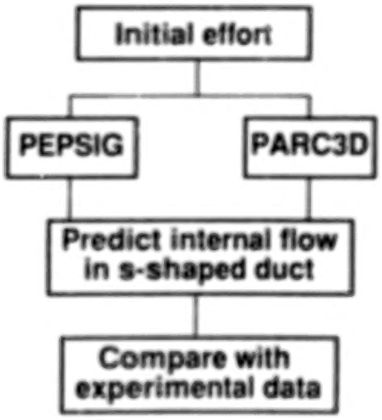
The effect of either opening the auxiliary inlets or drooping the cowl lip on inlet performance over a range of angles-of-attack from 0 to 110° at a free-stream Mach number of 0.12 is shown in this figure from reference 3. Performance is given in terms of recovery (engine face average total pressure divided by free-stream total pressure) and distortion (engine face maximum total pressure minus engine face minimum total pressure divided by engine face average total pressure). Total pressure distortion values above 15 percent are generally considered unacceptably high for initial screening purposes.

High performance at low subsonic speed/high angles-of-attack can be achieved by either opening the auxiliary inlets or drooping the cowl lip. The best performance was achieved with the 70° drooped cowl lip. Recovery was high, above 96 percent for angles-of-attack up to 100°, and distortion was low at 8 percent. If the cowl lip was not drooped (or the auxiliary inlets were not opened), the inlet would be limited to angles-of-attack below about 20° to stay within the acceptable level of distortion.

In summary, the data base contains only a limited number of steady-state pressure measurements obtained on subscale isolated inlet configurations. Consequently, the data base is not yet sufficiently complete for evaluating computer codes. However, the data base will be extremely useful in defining inlet configurations and techniques that have the potential for achieving high performance at low subsonic speed/high angle-of-attack conditions.

Computational Effort

- **A key goal is to predict internal performance of installed inlets at high alpha**
- **Use internal/external flow codes since internal performance effected by external flow**



Initial effort

PEPSIG

PARC3D

Predict internal flow in s-shaped duct

Compare with experimental data

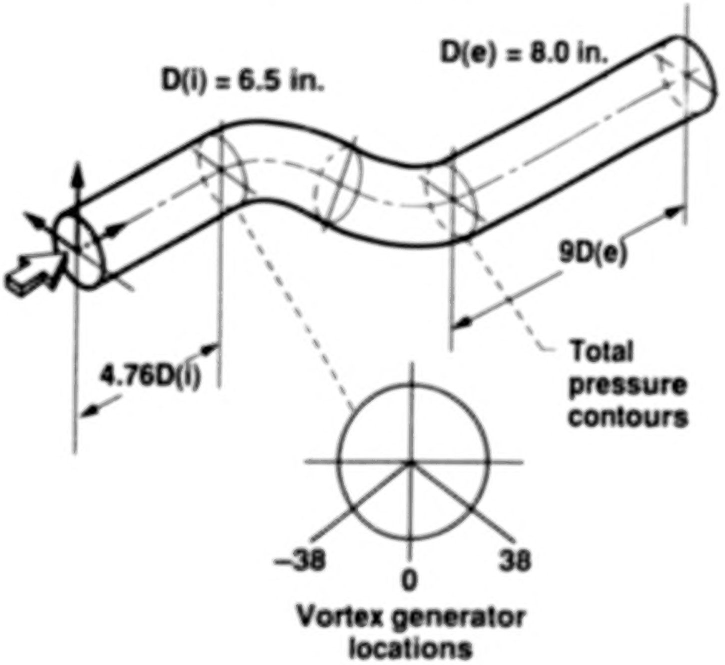
CD-91-54032

One of the key goals of this program is to calibrate/validate computer codes that can be used to predict the internal performance of installed inlets at high angle-of-attack conditions. Internal performance of installed inlets will be influenced by the external flow field upstream of the inlet. Thus, it will be necessary to use both internal as well as external flow codes. As a first step in the process, Lewis has been concentrating on two codes: PARC3D (ref. 4), the full Navier-Stokes code which can be used to predict external as well as internal flow fields, and PEPSIG (ref. 5), a parabolized Navier-Stokes code which can be used to predict internal flow fields.

Lewis computational capability will be illustrated using PARC3D and PEPSIG codes to predict the complex flow in an S-shaped diffusing duct. Predicted results are compared with experimental data taken from reference 7.

Computational Effort (continued)
Diffusing S–Duct

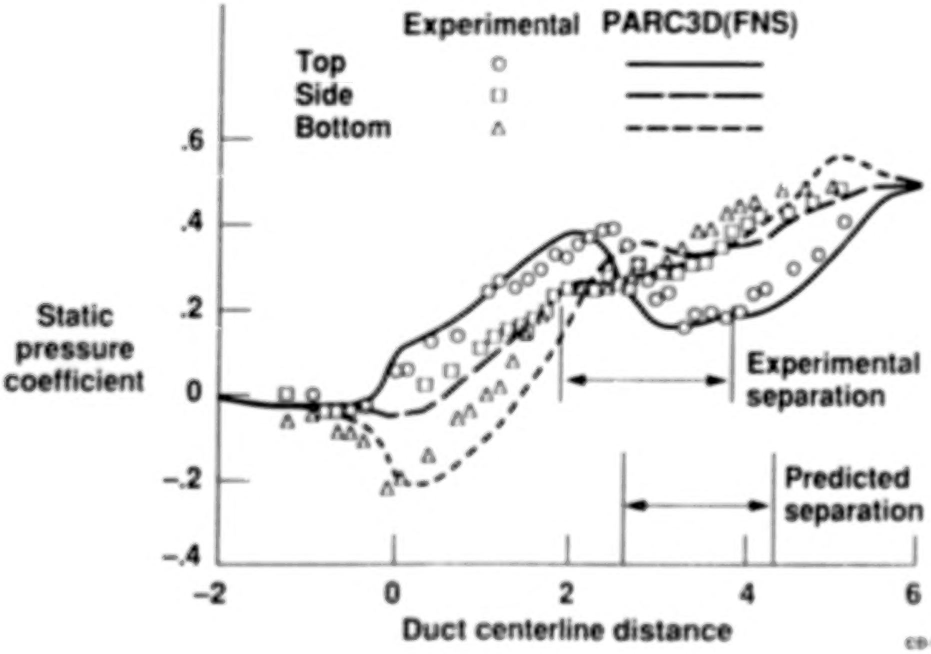
A(exit)/A(inlet) = 1.5; M(inlet) = 0.6



CD-91-54033

A sketch of the S-shaped diffusing duct is shown here. It has a circular cross section with two 30° bends resulting in a vertical offset of about one inlet duct diameter. The duct has an entrance diameter of 6.5 in. and an exit diameter of 8 in. resulting in a diffusing area ratio of 1.5. Inlet Mach number was 0.60. Flow separation and reattachment occurred in the duct, which also had strong secondary flow similar to that observed in some aircraft inlets. For some tests, three pairs of counter-rotating circular arc vortex generators were installed in the duct just downstream of the entrance near the bottom of the duct at circumferential angles of -38, 0, and 38°, as shown in the figure. The vortex generators were set at an angle-of-attack of 16° to the axial direction.

Computational Effort (continued)
Surface Static Pressure Distribution
No Vortex Generators

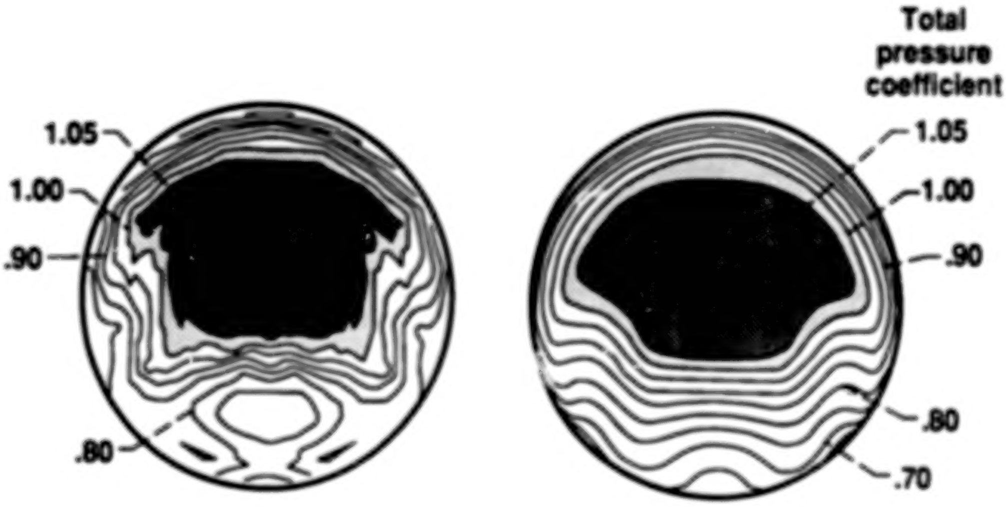


CD-91-54234

The predicted surface static pressure distribution from the PARC3D code at three circumferential locations is compared to the experimentally measured distribution in this figure in terms of the static pressure coefficient (local surface static pressure minus free-stream static pressure divided by free-stream dynamic pressure). Vortex generators were not installed and, consequently, flow separation occurred in this duct. Predicted flow separation occurred about one-half of a duct diameter downstream of the actual separation location. Also, the predicted static pressure distribution did not agree very well with the experimental distribution.

Three factors contribute to this discrepancy. Two of them, grid resolution and turbulence modeling, can affect the generation of vorticity and viscosity respectively. The third factor is the large tolerance of the internal surface of the duct walls. This factor resulted in some waviness of the internal surface of the experimental model compared to the smooth internal surface of the computational model. This data base, however, is one of the few available that contains very useful information on high subsonic flows in S-ducts.

Computational Effort (concluded)
Total Pressure Contours With Vortex Generators



Experimental

PEPSIG (PNS)

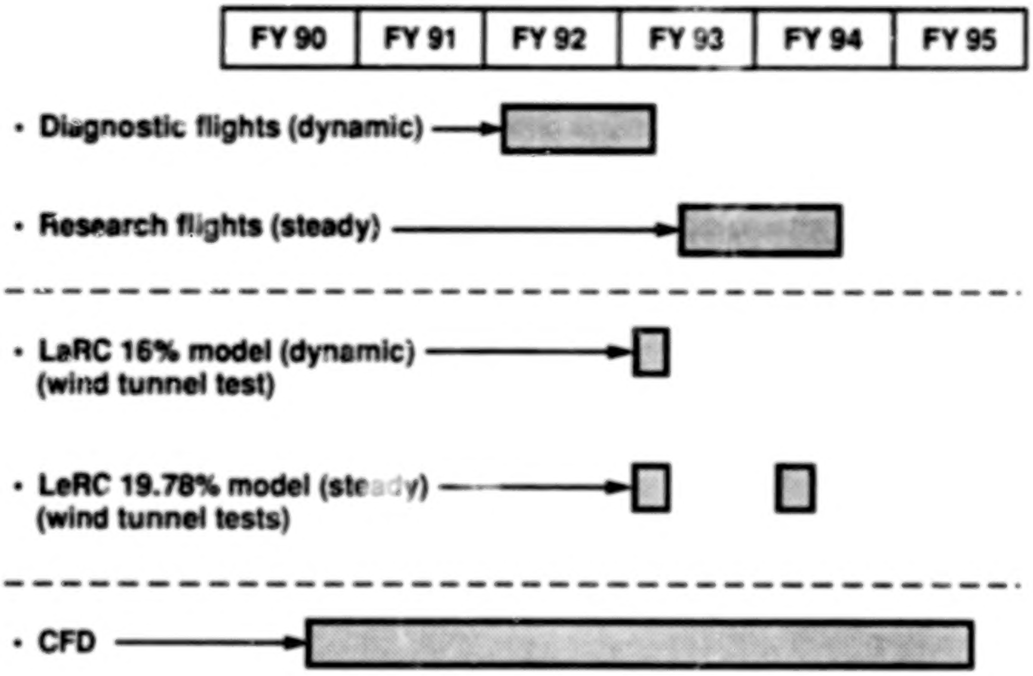
CD-81-[illegible]

In addition to a static pressure distribution, a comparison of total pressures was made and is shown in this figure for installed vortex generators. The predicted total pressure contour at the duct exit is compared with the experimental contour in terms of a total pressure coefficient. (Total pressure coefficient is defined as the local total pressure minus the free-stream static pressure divided by the free-stream dynamic pressure.) Predicted results were obtained using the PEPSIG code, which models the vortex generators in terms of a vorticity signature calculated from the physical properties of the generators themselves. Concentration of low total pressure near the bottom of the duct and high total pressure near the top of the duct is typical of that in S-ducts.

The agreement between predicted and actual total pressure contours is fairly good. However, the predicted region of low energy flow is somewhat smaller and the high energy flow region is somewhat larger than that of the experimental contour. Also, the predicted counter-rotating vortices near the bottom of the duct are less evident than in the actual flow. These discrepancies could be due to grid resolution, turbulence modeling and/or vortex generator modeling which can affect the generation of vorticity and viscosity.

In summary, these results are encouraging. There are, however, difficulties associated with the prediction of turbulent three-dimensional flows that contain strong secondary flows as well as separated flows. Further studies using denser grids as well as higher order turbulence and vortex generator models may improve these predictions.

Future Plans — Inlets



CD-91-[illegible]

Future plans for inlet tests are divided into the three categories of F-18 HARV flight tests, subscale tests, and computational effort (CFD), as shown in the figure. Flight tests are subdivided into diagnostic and research flights. Diagnostic flights, scheduled for about 1 year starting in the first quarter of FY 92, will investigate why F-18 aircraft experienced thrust losses when performing dynamic maneuvers outside the normal flight envelope during the Navy envelope expansion program. This effort will be followed by research flights also lasting about 1 year, which will generate a limited inlet steady-state data base for code evaluation.

Subscale model tests will be conducted at NASA Langley and Lewis. A 16-percent scale model F-18 HARV scheduled for a 6-week test in the Langley 14- by 22-ft tunnel, starting about the first quarter of FY 93, will investigate the effect of pitch and yaw rates on inlet flow field and inlet performance. The results will be compared with data from the diagnostic flights. A 19.78-percent scale model of the F-18 HARV forebody/inlet scheduled for two 6-week tests in the Lewis 9- by 15-ft tunnel (the first in FY 93 and the second in FY 94) will generate an inlet steady-state data base for code evaluation. These results will be compared with the more limited data base from the research flights.

The computational effort has already started and will continue into FY 95. It will be directed principally towards predicting the internal performance of the installed HARV inlet at high angles-of-attack. The codes will be evaluated and improved using the experimental data base generated from both flight and subscale model tests.

CONCLUDING REMARKS

This paper has described the existing Lewis technology for achieving high inlet performance at low subsonic speed/high angles-of-attack conditions and the plans to extend this technology to advanced highly maneuverable aircraft. Both the existing technology and the future plans include experimental as well as computational parts.

The experimental part of the existing technology consists of a data base generated over the past 20 years primarily in support of subsonic and supersonic VSTOL and STOVL aircraft at low subsonic speed/high angle-of-attack conditions, and contains results principally from subscale, isolated inlet configurations. Attractive techniques for achieving high performance at low subsonic speed/high angle-of-attack conditions have been shown. The data base, however, does not contain sufficient information for a complete evaluation of computer codes.

The computational part of the existing technology consists of results principally from studies using an S-shaped diffusing duct with and without vortex generators. Flow separation and reattachment occurred in this duct, which also had a strong secondary flow. The full Navier-Stokes code, PARC3D, and a parabolized Navier-Stokes code, PEPSIG, were used to predict the complex flow field within the duct. The results are encouraging. Further studies with denser grids as well as the use of higher order turbulence and vortex generator models may improve the prediction capability.

Future plans include both experimental and computational efforts applicable to external and internal flows associated with the F-18 HARV as well as with a subscale F-18 model at low subsonic speed/high angle-of-attack conditions. The experimental effort will be directed principally towards expanding the existing data base to include detailed information on forebody/inlet flow field interaction and its effect on inlet performance. The computational effort will be directed principally towards predicting the internal performance of the installed HARV inlet at high angles-of-attack. The codes will be evaluated and improved using the experimental data base generated from both flight and subscale model tests.

REFERENCES

1. Mitchell, G.A.: Effect of Inlet Ingestion of a Wing Tip Vortex on Turbojet Stall Margin. NASA TM X-71610, 1974.

2. Luidens, R.W.; Stockman, N.O.; and Diedrich, J.H.: Optimum Subsonic, High Angle-of-Attack Nacelles. 12th Congress of the International Council of the Aeronautical Sciences, J. Singer and R. Staufenbiel, eds., AIAA, 1980, pp. 530-541 (also, NASA TM-81491).

3. Mark, J.L.; McGarry, M.A.; and Reagan, P.V.: Research on a Two-Dimensional Inlet for a Supersonic V/STOL Propulsion System. NASA CR-174945, 1984.

4. Cooper, G.K.: The Parc Code: Theory and Usage. AEDC-TR-87-24, Oct. 1987 (avail. NTIS, AD-B115169).

5. Briley, W.R.; and McDonald, H.: Three-Dimensional Viscous Flows with Large Secondary Velocity. J. Fluid Mech., vol. 144, Mar. 1984, pp. 47-77.

6. Thomas, J.L., et al.: A Patched-Grid Algorithm for Complex Configurations Directed Towards the F-18 Aircraft. AIAA Paper 89-0121, Jan. 1989.

7. Vakili, A.D., et al.: Compressible Flow in a Diffusing S-Duct with Flow Separation. Heat Transfer and Fluid Flow in Rotating Machinery, W.J. Yang, ed., Hemisphere Publishing Corp., Washington, D.C., 1985, pp. 201-211.

N91-20092

OVERVIEW OF HYPERSONIC/TRANSATMOSPHERIC VEHICLE PROPULSION TECHNOLOGY

John E. Rohde
NASA Lewis Research Center
Cleveland, Ohio

Significant progress is being made in attaining many of the enabling technologies critical to the development of future hypersonic vehicles such as the single-stage-to-orbit National Aerospace Plane (NASP) X-30 vehicle and others. In the NASP program, we have safely produced and transferred slush hydrogen in modestly large quantities. Inlet testing has demonstrated that a high-performance configuration can be developed, and configurations have been developed that reduce cross-talk between engine modules and improve unstart margins. Tests of a hydrogen-fueled ramjet engine model have been conducted to investigate engine operability and dynamics, and future tests are planned to demonstrate a closed-loop engine control system. Nozzle tests have identified large transonic-drag losses in the nozzle, which were then reduced significantly through the use of external burning. In other research areas, such as engine seals, advanced materials, and para-to-ortho hydrogen conversion, promising concepts have been identified and continuing efforts are adding to these technologies. In the Generic Hypersonics program we are gaining an improved understanding of the physics of inlets, combustors, and nozzles and are developing advanced materials and computational codes that predict the characteristics of both reacting flows and metal-matrix composite structures. The High-Mach Turbine-Engine (HiMaTE) component technology program has identified the turboramjet and the air-turboramjet as the most promising combined-cycle engines for more detailed assessment. Efforts are continuing to define critical components for development and testing to demonstrate technology readiness and to establish cycle feasibility. Studies are being conducted to assess the potential benefit of using these combined-cycle engines to power the first stage of two-stage-to-orbit (TSTO) vehicles and other hypersonic vehicles.

Hypersonic/Transatmospheric Vehicle Propulsion Technology

Focused Technology Programs

- National Aerospace Plane
- Generic Hypersonics

Fundamental R&T

- Inlets
- Nozzles/External burning
- Engines/Controls
- Slush hydrogen
- Para-to-ortho hydrogen
- Sensors
- Seals
- Materials
- Analyses codes
  - Fluid dynamics
  - Structures

- High-Mach Turbine Engine (HiMaTE) (Joint NASA-Air Force)
  - Cycle studies
  - Component technology demonstrations
  - Possible combined-cycle demonstration

System studies

- TSTO vehicles
- Advanced missiles
- Mach-4 to -6 aircraft

CD-91-54489

The Hypersonic/Transatmospheric Vehicle (TAV) propulsion technology effort at NASA Lewis Research Center encompasses three focused technology programs: the National Aerospace Plane (NASP) with its single-stage-to-orbit (SSTO) X-30 vehicle, NASA's Generic Hypersonics, and the joint NASA-Air Force High-Mach Turbine-Engine (HiMaTE) component technology program. The NASP Program focuses primarily on scramjet propulsion systems which are highly integrated with SSTO vehicles. NASP must rely on near-term technologies because of an early first flight. Generic Hypersonics, on the other hand, focuses on the development of fundamental research and technology required for the design of future air-breathing propulsion systems to power accelerating and other hypersonic vehicles. This program focuses on longer-term technologies. The HiMaTE component technology program focuses on turbomachinery-based, air-breathing propulsion systems for transatmospheric and hypersonic vehicles operating in the Mach 4 to 6 flight speed range. This program will result in the development of nearer-term technologies which can enable nearer-term air-breathing propulsion vehicles.

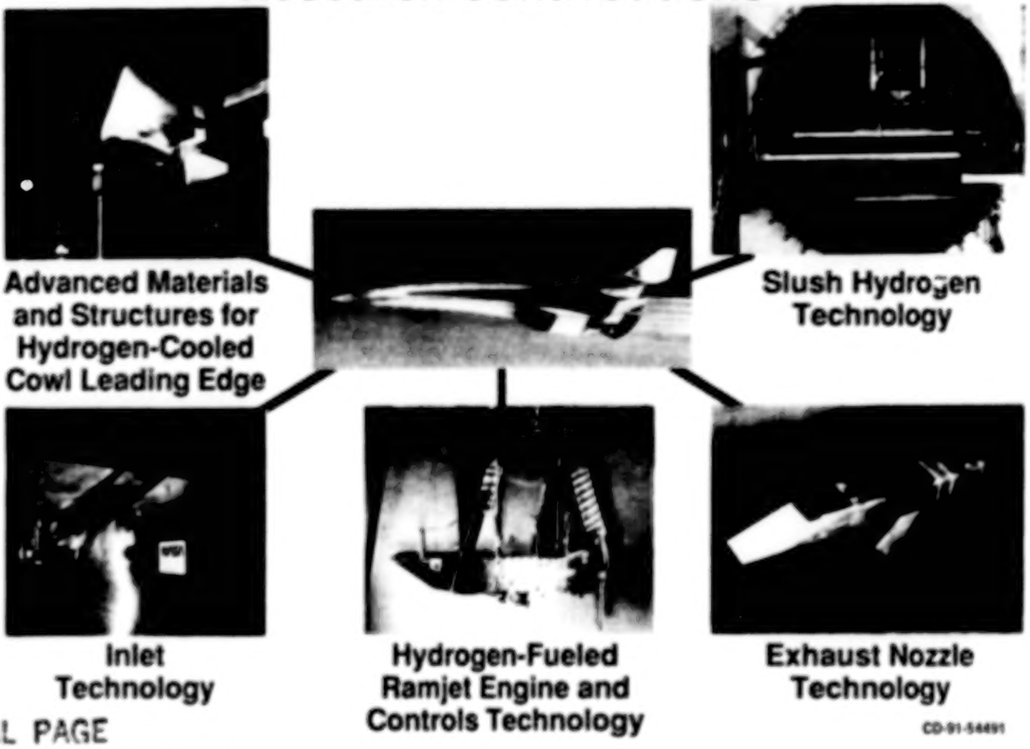
Hypersonic/Transatmospheric Vehicle PropulsionTechnology
(concluded)

Goals

- Provide enabling hypersonic technology for future hypersonic/transatmospheric flight vehicles
  - Place emphasis on air-breathing propulsion
  - Blend technologies from aeronautics and space propulsion

CD-91-54490

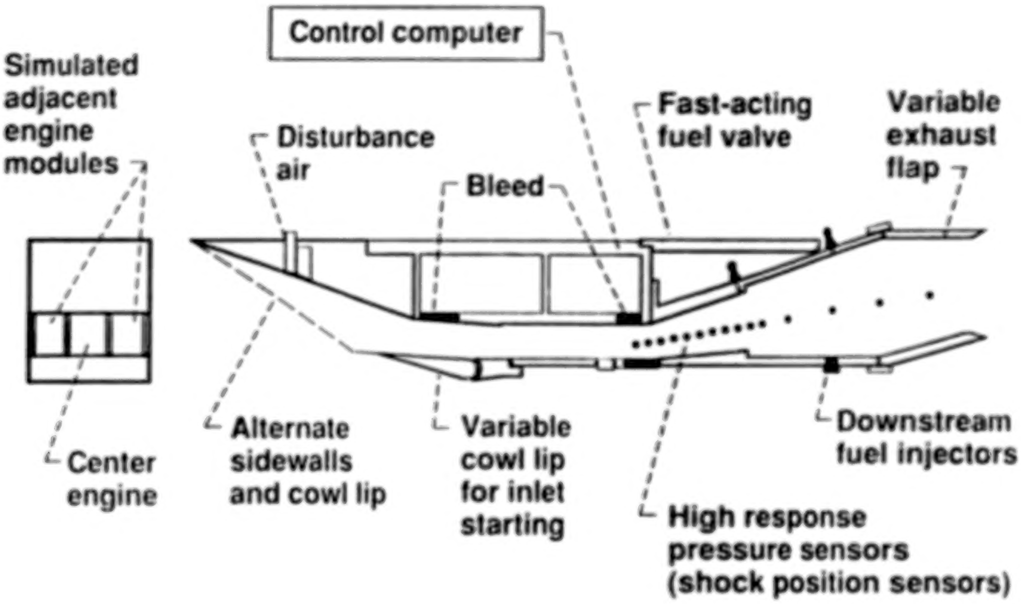
Hypersonic/Transatmospheric Vehicle Research Contributions


Advanced Materials and Structures for Hydrogen-Cooled Cowl Leading Edge
Slush Hydrogen Technology
Inlet Technology
Hydrogen-Fueled Ramjet Engine and Controls Technology
Exhaust Nozzle Technology
CD-91-54491

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Currently the NASA Lewis Research Center is conducting propulsion system research which will be used to develop the National Aerospace Plane and other transatmospheric and hypersonic vehicles. Significant contributions are being made in the following areas: advanced materials and structures for hydrogen-cooled engine components and seals; wind tunnel tests for inlet studies, operational characteristics of hydrogen-fueled ramjets, and exhaust nozzles; and slush-hydrogen fuel production, use, storage, and handling. Lewis is also supporting NASP by developing advanced para-to-ortho hydrogen conversion catalysts that will enhance the cooling capability of hydrogen at cryogenic temperatures. Finally, analytical codes are being developed and used extensively to predict the high-speed internal flows in components and engine systems, the structural characteristics of metal-matrix composites, and the characteristics of slush hydrogen in fuel and storage tanks.

Modified Government Baseline Engine For NASP


Control computer
Simulated adjacent engine modules
Disturbance air
Bleed
Fast-acting fuel valve
Variable exhaust flap
Center engine
Alternate sidewalls and cowl lip
Variable cowl lip for inlet starting
Downstream fuel injectors
High response pressure sensors (shock position sensors)

CD 91 54492

A hydrogen-fueled ramjet engine model, referred to as the NASP government baseline engine, was recently modified extensively to investigate engine operability and control. The model now incorporates all of the major NASP engine features as well as several design options and special instrumentation. For simplicity and reduced cost, the subscale engine model was built using heat-sink construction, which limits the duration of tests. Some of the major engine features include bleeds in the inlet and throat regions; a variable cowl lip for inlet starting, inlet flow passages on either side of the center engine to simulate adjacent engine modules and study module-to-module interactions; alternate sidewall configurations designed to reduce or eliminate adverse engine interactions; and the capability to induce high-frequency disturbances of the inlet and fuel flows for dynamic tests. A 6-month test of this engine model was conducted by Lewis at the General Applied Science Laboratories (GASL) to investigate engine operability, dynamics, and performance at a Mach-3.5 flight condition.

Modified Government Baseline Engine Test Results Summary

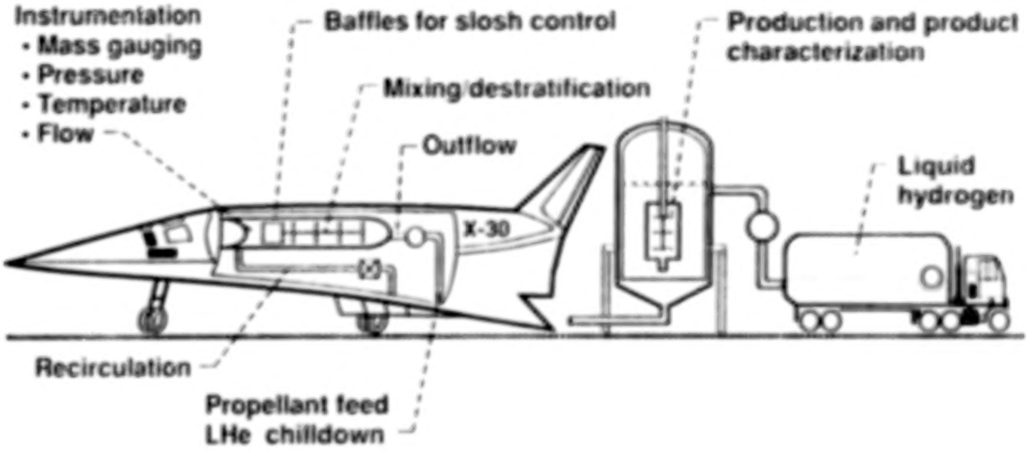
- Variable cowl lip permits inlet starting and provides good performance
- Combustor ignition achieved with spark igniter (silane not required)
- Representative combustor operating region defined for dynamic tests
  - High equivalence ratios reached without inlet unstart
- Engine relight and inlet restart sequences demonstrated
- Engine dynamic response to inlet and fuel flow disturbances measured for comparison to analysis and input to control design

CD 91 54003

Improved understanding of engine operability and other significant results was obtained in recently completed tests of the engine at GASL. We used these results to define an inlet configuration which has both excellent starting-restarting characteristics and high performance levels. During the combustor testing, ignition was achieved by using only a spark ignitor, and stable combustor operation was achieved at the desired high-pressure conditions without unstarting the inlet (the equivalence ratio was limited to about 0.8 by facility interactions). Unique operating procedures, which included both an inlet and combustor restart sequence, were developed to both start and relight the engine. The effects of an adjacent engine module unstart on engine operation were experimentally evaluated at various operating conditions, and configurations were developed to reduce cross-talk between engine modules and to improve the unstart margin. The engine dynamic response to inlet and fuel flow disturbances was measured for comparison to the analyses. This dynamic response information is now being used to modify the design of a control system which will provide closed-loop engine control in an upcoming test in the Lewis Propulsion Systems Laboratories (PSL). These test results are helping designers to develop the NASP engine flow path and control system.

Slush-Hydrogen Technology

Technology Advances Needed: **Ground Handling**

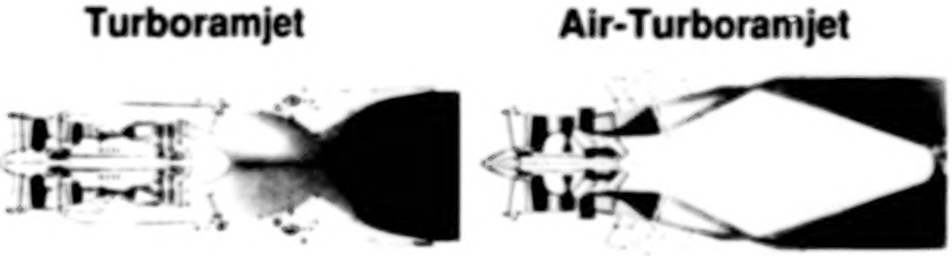

Instrumentation
• Mass gauging
• Pressure
• Temperature
• Flow
Baffles for slosh control
Mixing/destratification
Outflow
Production and product characterization
Liquid hydrogen
X-30
Recirculation
Propellant feed
LHe chilldown

Benefits of slush hydrogen
- 16 % increase in density
- 18 % increase in heat-sink capacity
- Reduced tank pressure (conformal tanks)

→ Decreased aircraft volume and drag → 15 to 35% lower takeoff weight

Slush hydrogen as a fuel is being investigated to support the NASP program. A viable slush-hydrogen fuel would contain a mixture of approximately 50 percent solid hydrogen suspended in the liquid hydrogen. This mixture would provide a 16-percent increase in propellant density and an 18-percent increase in heat-sink capacity, which translates directly to decreased vehicle volume and drag. The resulting major reduction in vehicle gross takeoff weight is from 15 to 35 percent, depending on whether or not warm hydrogen is recirculated to the tank. The critical slush-hydrogen technologies are related to making, storing, recirculating, mixing, transferring, measuring, and predicting characteristics. We are investigating these technologies at our K-Site facility at Plum Brook Station. Recently, a slush mixture with a solid-to-liquid fraction of 60 to 65 percent was produced by the freeze-thaw method, successfully transferred for the first time to a 500-gallon research tank, and subsequently expelled from the tank. These tests clearly demonstrated that slush hydrogen can be produced and safely distributed in modest quantities according to the NASP X-30 requirements. In conjunction with the testing, analytical codes are being developed to help researchers understand and predict the flow characteristics and fluid properties inside fuel and storage tanks at various operating conditions.

HiMaTe Component Technology Propulsion Program



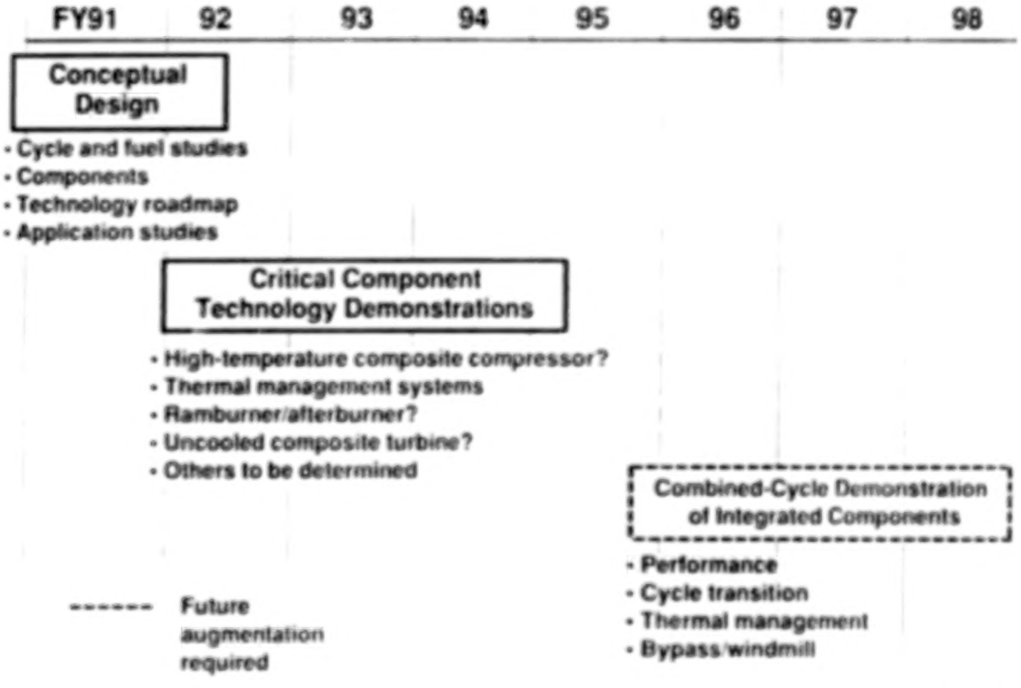
Objectives:

- Investigate propulsion advantages of combining the best air-breathing and rocket technologies in an engine capable of operating over a range of flight conditions from Mach 0 to 6
- Conduct critical component demonstrations to establish cycle feasibility and technology readiness

CD-91-54695

The HiMaTE component technology program attempts to blend the best features of both air-breathing and rocket propulsion technologies in one combined-cycle engine system capable of operating over the complete flight speed range from Mach 0 to 6. Future hypersonic vehicles will require a propulsion system which provides both high performance and high thrust-to-weight ratios while operating in a very severe thermal environment. High-performance turbomachinery will be pushed to as high a speed condition as possible (to about Mach 3.5+). Using rocket technology, a gas generator will be considered to power the turbine; thus, operations at high-speed conditions will not be limited by the hot section of the engine because of high inlet air temperatures. Using a gas generator will also permit a reduced turbine size, which will significantly increase the thrust-to-weight ratio of the engine. At the higher speed conditions (from Mach 4 to 6), a ramjet cycle will be used for high performance, and regenerative fuel cooling will be used to enable adequate thermal management without compromising performance. The turboramjet and airturboramjet have been identified as the most promising combined-cycle engines for more detailed assessment. At the higher speed conditions, these engine systems, respectively, bypass the flow around the turbomachinery and windmill the turbomachinery.

HiMaTE Component Technology Program (Joint NASA--Air Force Program)



Conceptual Design

- Cycle and fuel studies
- Components
- Technology roadmap
- Application studies

Critical Component Technology Demonstrations

- High-temperature composite compressor?
- Thermal management systems
- Ramburner/afterburner?
- Uncooled composite turbine?
- Others to be determined

Combined-Cycle Demonstration of Integrated Components

- Performance
- Cycle transition
- Thermal management
- Bypass/windmill

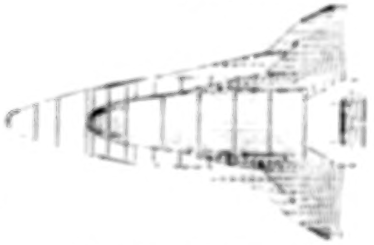
------ Future augmentation required

CD-91-54486

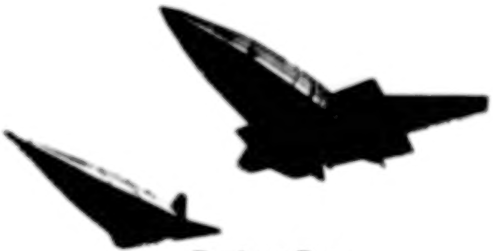
General Electric Company and Pratt & Whitney are the two HiMaTE prime contractors working on the turboramjet and air-turboramjet combined-cycle propulsion systems, respectively. Currently, they are conducting conceptual design studies to assess the cycles, fuels, and potential benefits of these propulsion systems in various applications. Also engines and individual components are being designed and the component technology needs assessed relative to the state-of-the-art. The assessment includes choosing materials and laying out technology roadmaps to validate component technologies and cycles. Shortly, the contractors in concert with NASA and the Air Force will select unique, critical components and identify those efforts required to develop and demonstrate component technology readiness. As yet, all of these critical components have not been identified, but a thermal management system, which is required to operate in the severe thermal flight environment of Mach 4 to 6, will be one of the required unique elements. Other critical component technologies being considered are a high-temperature composite compressor, a ramburner/afterburner, and an uncooled composite turbine. After the selected critical components have been demonstrated, a funding augmentation will be advocated for conducting a combined-cycle demonstration in the Lewis PSL facility. This demonstration of the components integrated into a system would be used to verify performance levels, cycle transition, thermal management, and hot gas bypass or windmilling of the turbomachinery.

Transatmospheric Vehicle Studies

Objective: demonstrate potential of utilizing turbomachinery-based propulsion concepts in transatmospheric vehicles

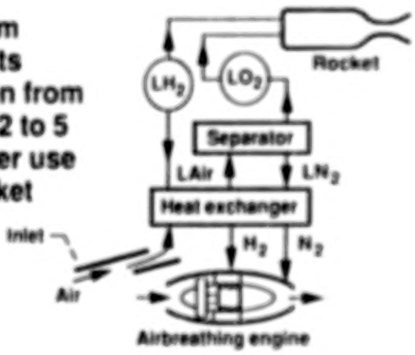


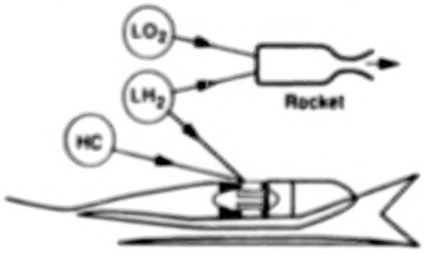
Air Liquefaction and Enrichment System (ALES) (NASA/Air Force/Rockwell)



Project Beta (NASA/Air Force/Boeing)

- **System collects oxygen from Mach 2 to 5 for later use in rocket**





Over/Under Turboramjet

CD-91-54497

Transatmospheric vehicles (TAV's) and other military vehicles such as a multirole Mach-4 to -6 aircraft and advanced missiles are being studied to evaluate the potential benefits from using turbomachinery-based air-breathing propulsion systems in these applications and to define the propulsion requirements. Two studies are being conducted on two-stage TAV's to evaluate the potential for using low risk air-breathing propulsion to power the first-stage booster and rockets to power the orbiter. In these two-stage-to-orbit (TSTO) vehicle applications, the air-breathing propulsion system weight for the booster is not as critical a driving parameter as it is in the single-stage-to-orbit (SSTO) vehicles because the booster is only accelerated to Mach 5 or 6. The first study, Project Beta is a joint effort with the Air Force and Boeing to evaluate a two-stage vehicle with a bottom-mounted orbiter capable of putting a 10 000-pound payload into polar orbit. Current study vehicles use near-term materials and a hydrocarbon (HC)- and hydrogen-fueled over-under turboramjet engine to power the booster vehicle to Mach 6, followed by a staging of the rocket-powered orbiter. The second study uses an air-liquefaction and enrichment system (ALES). This study is a joint effort with the Air Force and Rockwell to evaluate a two-stage vehicle with a top-mounted orbiter. This vehicle is also capable of putting a 10 000-pound payload into polar orbit. Current study vehicles use near-term materials and a hydrogen fueled air-turboramjet propulsion system for powering the booster up to about Mach 5, where the rocket-powered orbiter is then launched. During first-stage operation of the air-breathing propulsion system, large quantities (at least 30 percent) of air are liquified, separated, and stored in the orbiter for later use in the second-stage rocket. These two-stage-to-orbit vehicles using air-breathing propulsion for the first stage offer a lower risk approach which still presents major engineering challenges but does not require the invention of a major new propulsion system.

Summary

These focused propulsion technology efforts will

- Extend efficient air-breathing engine operation to Mach 6 and provide high thrust-to-weight engines
- Enable the development of cruise vehicles capable of higher speeds (Mach 4 to 6)
- Enable development of two-stage-to-orbit vehicles with low risk air-breathing propulsion

CD 91-[illegible]

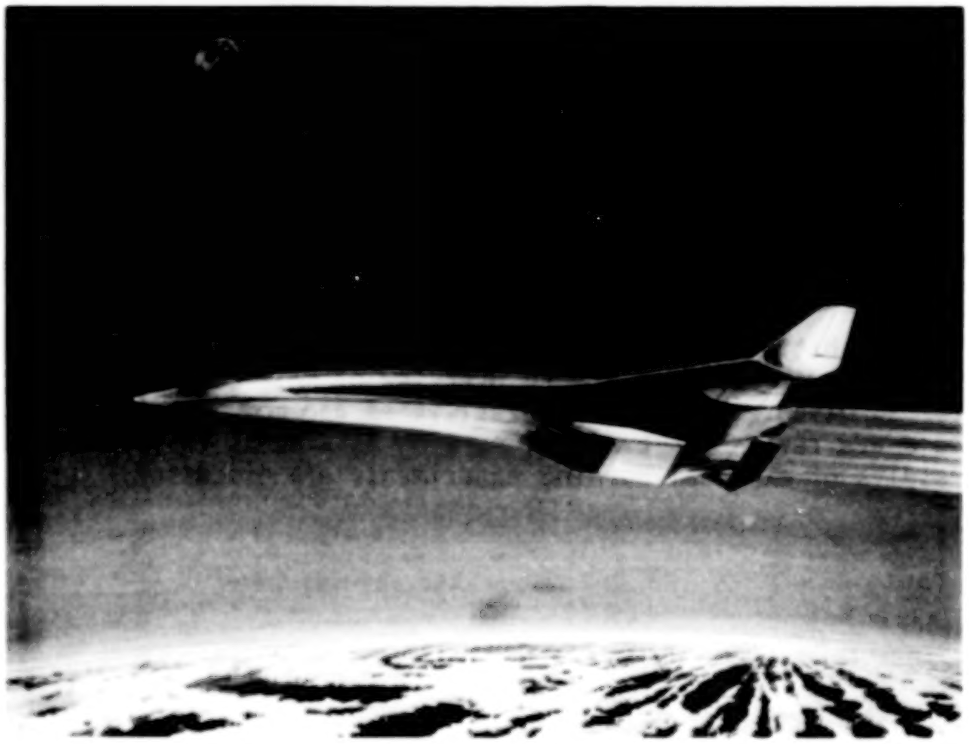
N91-20093

OVERVIEW OF NASP NOZZLE RESEARCH

Charles J. Trefny, Rickey J. Shyne, and Jeanne D. Carboni
NASA Lewis Research Center
Cleveland, Ohio

Problems facing the National Aerospace Plane (NASP) nozzle at transonic and low supersonic conditions are discussed. An overview of NASP nozzle research at Lewis Research Center is given. Experimental facilities and computational techniques currently in use are reviewed. External burning as a means to reduce transonic drag and initial results of external burning experiments are discussed.

Aerospace Vehicle Employing Highly–Integrated Exhaust Nozzle



CD-91-54349

**ORIGINAL PAGE IS
OF POOR QUALITY**

Interest in transatmospheric or aerospace vehicles has been revived in the United States following almost two decades of relative inactivity. Evolutionary advances in scramjet propulsion, materials, and computer modeling have set the stage for an aggressive program (the National Aerospace Plane, or NASP) to develop a revolutionary aircraft capable of flying into orbit following takeoff from a conventional runway. Ready access to space, and very high speed Earth transportation are but two of the obvious benefits of this technology. A single stage to orbit concept is very attractive due to its operational simplicity, flexibility and its potential for reducing the cost of putting payload into orbit. The technical challenges facing the aerospace community are numerous, many of them related to the airbreathing propulsion system required to achieve orbit in a single stage. Liquid hydrogen fuel is widely accepted as the fuel of choice for hypersonic airbreathing propulsion due to its high heat capacity for engine and airframe cooling, and a heat of combustion twice that of hydrocarbon fuels. One drawback of hydrogen is its low molecular weight which results in a large cryogenic volume that must be highly integrated with the airframe and propulsion system. An artist's conception of the National Aerospace Plane appears in this figure to illustrate a typical highly integrated configuration with a large, scarfed, two-dimensional exhaust nozzle. The entire vehicle aft end acts as an expansion surface for the scarfed nozzle, providing a very high area ratio which is exploited at the high nozzle pressure ratios associated with high Mach number and altitude. This large aft-facing area becomes a critical issue, however, at transonic and supersonic speeds where relatively low airbreathing engine pressure ratios result in a highly overexpanded nozzle.

At Low Speed, the NASP Nozzle Is Highly Overexpanded

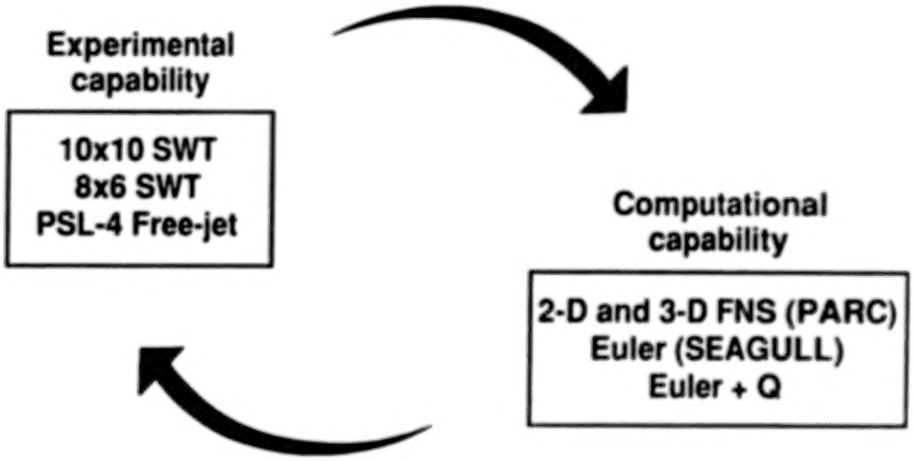
- **Determine nozzle operating characteristics from takeoff to moderate supersonic speed**
- **Develop drag-reduction techniques, specifically external burning**

CD-91-54350

The design of a nozzle required to operate from takeoff to orbit is a formidable task. The amount of variable geometry that can be employed is a small fraction of that required to keep a nozzle "on-design" over this speed range, mainly because of the invariability of the vehicle aft end which must be used as the nozzle expansion surface.

A primary goal of NASP nozzle studies at the NASA Lewis Research Center is to determine the performance characteristics from low to moderate supersonic speeds for the candidate configurations. The poor performance resulting from off-design operation necessitates some form of augmentation especially at transonic speeds. External burning has been studied at Lewis as a potentially attractive solution.

Approach



Experimental capability

10x10 SWT
8x6 SWT
PSL-4 Free-jet

Computational capability

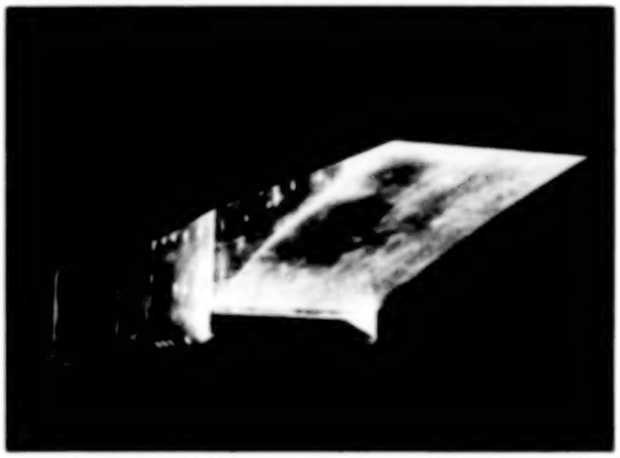
2-D and 3-D FNS (PARC)
Euler (SEAGULL)
Euler + Q

CD-91-54351

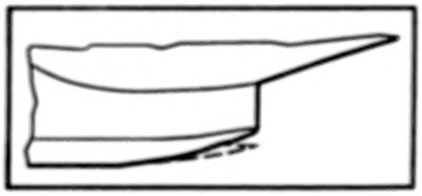
Nozzle research at Lewis encompasses a wide range of both experimental and computational activities. Recently, a heavier emphasis has been placed on building an hierarchy of numerical capability for performance prediction, wind tunnel model design, and experimental data analysis. Computer codes currently in use vary in sophistication from two-dimensional Euler analysis, such as SEAGULL, to three-dimensional Full Navier-Stokes (FNS) solvers, such as PARC. Experimentally, Lewis is using the 10- by 10-Foot and 8- by 6-Foot Supersonic Wind Tunnels (SWT's) as well as a number of smaller facilities such as a Mach-1.26 free-jet in cell 4 of the Propulsion Systems Lab (PSL-4).

NASP Jet-Effects Test Rig
Lewis 10x10 SWT

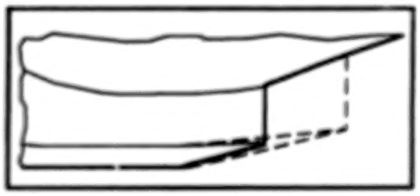
Installed in Lewis 10x10 SWT



Geometry Variations



Internal Area Ratio



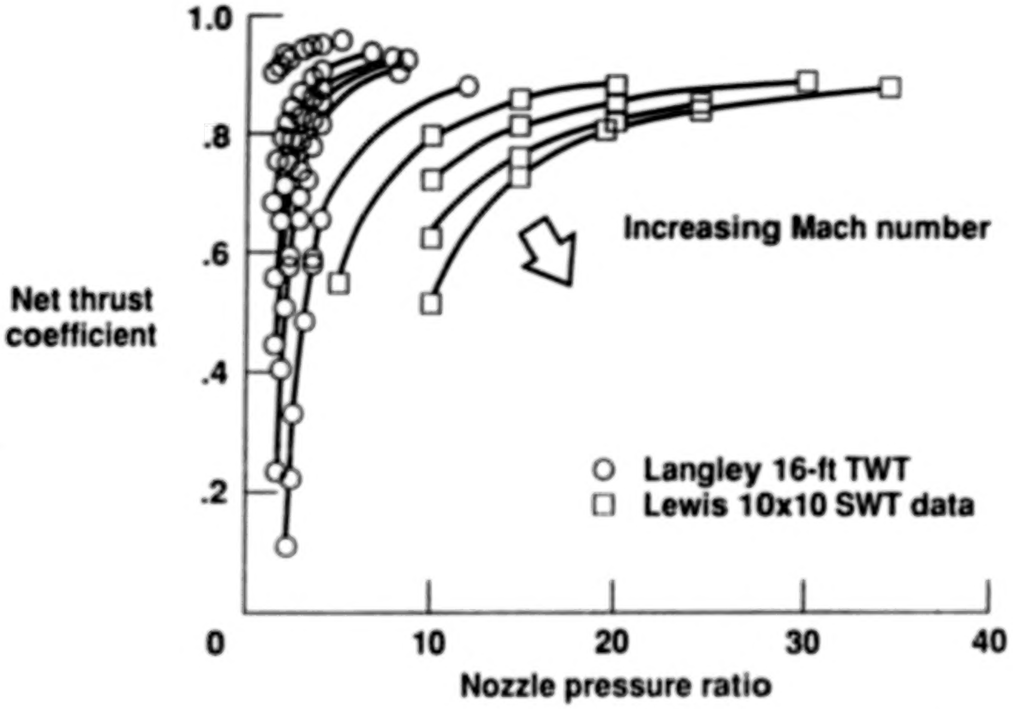
Extended Cowl

ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-54352

The primary purpose of this test was to determine the low-speed (Mach 2.0 to 3.5) performance of generic nozzle/afterbody configurations designed for a hypersonic propulsion system. This test was an extension of transonic tests conducted at the NASA Langley 16-Foot Transonic Wind Tunnel. The objectives of these tests were to determine nozzle afterbody thrust-minus-drag levels and vector angle, define separated flow regions on the afterbody with and without sidewalls, determine the influence of afterbody shape and length on performance, determine the effect of upstream contour on performance, provide an experimental database for computational fluid dynamics calibration, and determine throttle-dependent changes on afterbody forces. The data obtained from these series of tests establishes an experimental database for this class of nozzle/afterbody configurations and will be used to guide NASP propulsion/airframe design and integration.

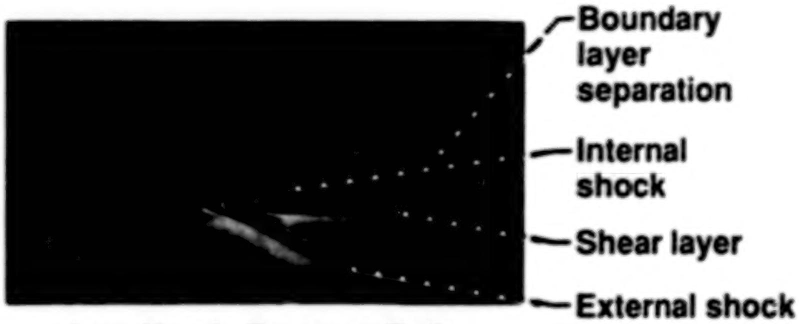
NASP Low-Speed Database Extended to Mach 3.5



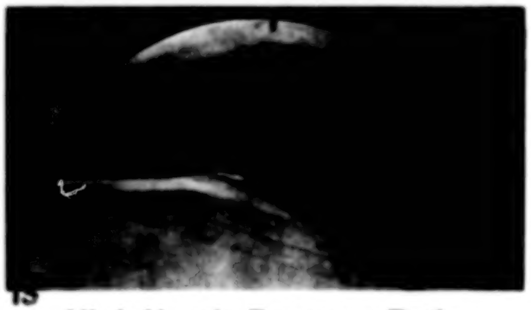
CD-91-54353

This is a plot of net thrust coefficient versus nozzle pressure ratio for the baseline nozzle/afterbody configuration. Transonic data (Mach 0 to 1.2) was obtained in the NASA Langley 16-Foot Transonic Wind Tunnel. Supersonic data (Mach 2 to 3.5) was obtained in the Lewis 10- by 10-Foot Supersonic Wind Tunnel. The net thrust shows an expected increase with nozzle pressure ratio, and the peak nozzle performance shifts toward higher nozzle pressure ratios as the Mach number increases. In general, the nozzle pressure ratio available in current NASP engine cycles is less than the peak value, necessitating thrust augmentation at low Mach numbers.

Schlieren Images Aid Understanding of Overexpanded Nozzle Flow Fields



Low Nozzle Pressure Ratio



ORIGINAL PAGE IS
OF POOR QUALITY

High Nozzle Pressure Ratio

CD-91-54354

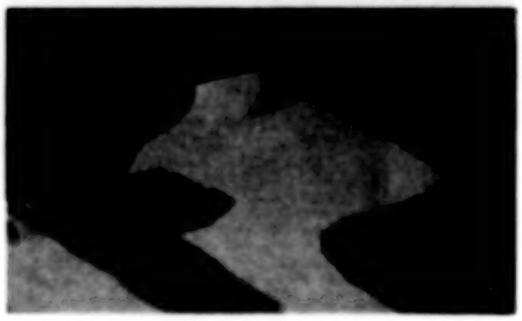
Schlieren flow visualization techniques were used to qualitatively evaluate the flowfields generated by an overexpanded nozzle in the Lewis 10- by 10-Foot Supersonic Wind Tunnel. The upper schlieren image shows an internal shock wave being generated by the nozzle/afterbody that intersects the nozzle ramp surface and causes the boundary layer to separate for a low nozzle pressure ratio, overexpanded flow condition. This phenomena is contrasted by the lower schlieren figure, which shows an external shock wave and shows the shear layer generated from the interactions of the nozzle plume and tunnel flow at a high nozzle pressure ratio, underexpanded flow condition. For this flow condition, the nozzle flows full without the boundary-layer separation caused by an internal shock wave, as in the underexpanded flow case.

Computational Capability for Overexpanded Hypersonic Nozzles

Schlieren Photo

ORIGINAL PAGE IS
OF POOR QUALITY

2-D Euler Solution

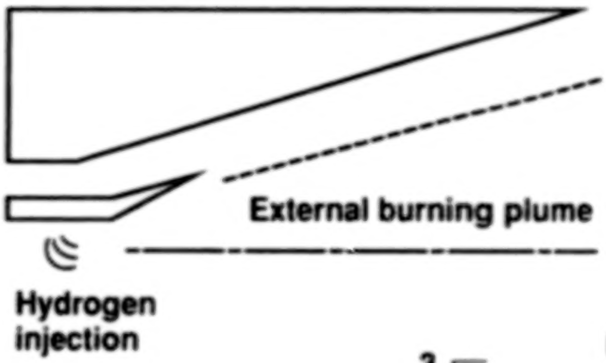


3-D Euler Solution

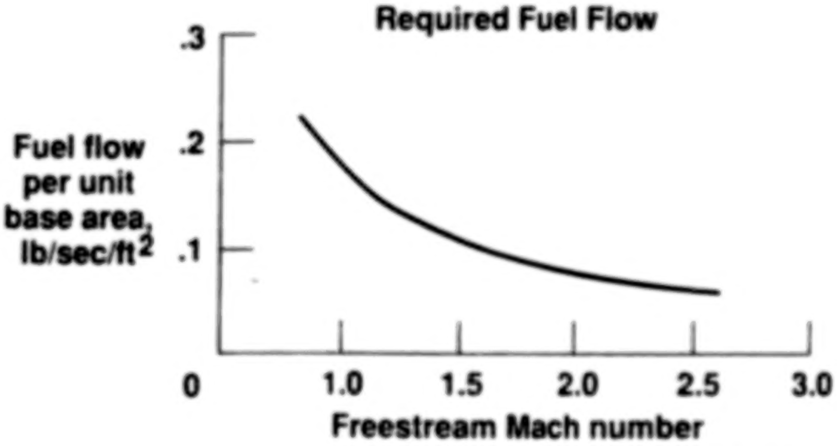
CD-91-54355

Computational analysis is necessary to predict and understand the behavior of the complex flowfields of hypersonic nozzles. Analytical models can provide insight into the characteristics of these flowfields, and they can also help predict performance of certain configurations - thus saving the cost associated with experimental testing. One way of comparing analytical data with experimental data is a density contour plot. Density levels can be qualitatively compared with schlieren photographs. On the left is a schlieren photograph taken of one configuration tested in the Lewis 10- by 10-Foot Supersonic Wind Tunnel. Here, the nozzle was operating at a pressure ratio of 15 in a Mach-2.5 free stream. Two computer analyses were used to model the same geometry, in an effort to predict the locations of the external shock, the shear layer, and the shock deflecting off the ramp surface - all seen clearly in the schlieren image. The top right figure shows results from SEAGULL, a two-dimensional Euler analysis. SEAGULL uses shock-fitting, a technique that calculates shock locations on the basis of discontinuities in pressure or geometry. The bottom right figure shows results from a PARC3D, three-dimensional Euler analysis. Both figures show favorable results in predicting the shear layer and shock locations.

External Burning to Reduce Transonic Drag



- Proof-of-concept experiments
- Control-volume analysis
- Euler analysis



CD-91-54356

A conceptually attractive solution to the transonic nozzle drag problem is the combustion of fuel in an external stream of air passing adjacent to the cowl. If the burning streamtube could be made to expand at free-stream pressure, then the drag on the cowl flap would be eliminated, and the exhaust flow would exit to ambient pressure without overexpanding. This concept was studied by using a control-volume approach to estimate the performance potential and fuel flow requirements. The fuel flow required to eliminate drag along a 1000-psfa dynamic pressure trajectory is shown. The roughly 0.1- to 0.2-lb/sec/ft^2 of base area gave high specific impulse performance and warranted further study.

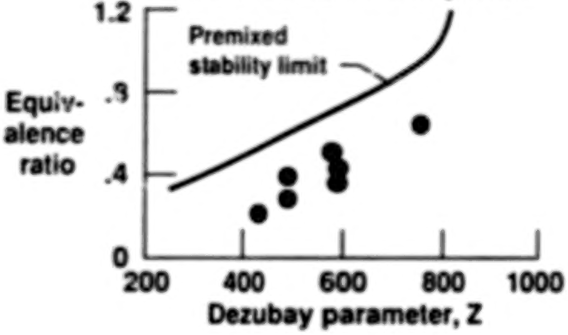
External Burning Flame Stability Experiment Helps Define Operational Limits

Spraybar Installed in PSL-4 Free-Jet

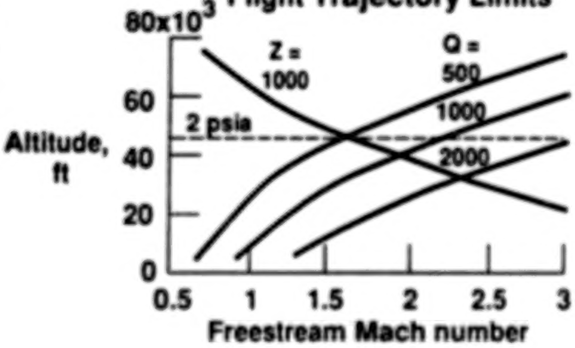
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54357

PSL-4 Basic Stability Data



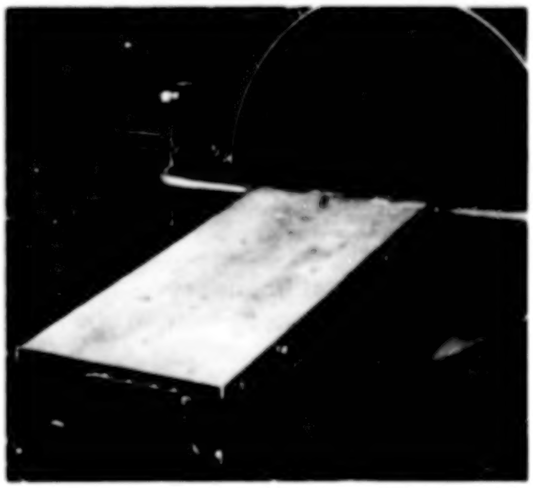
Flight Trajectory Limits

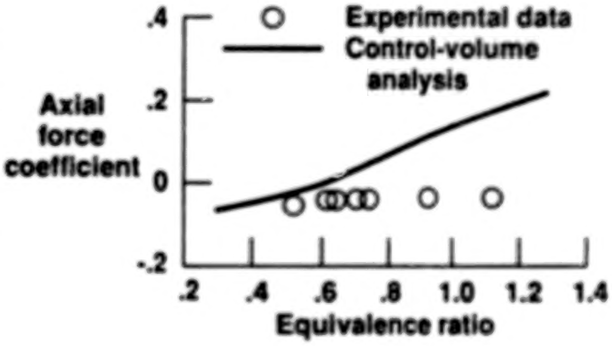


To determine combustion stability limits for a non-premixed stream of hydrogen and air, a spraybar test was conducted in a Mach-1.26 free-jet at Lewis. The photograph on the left shows the spraybar model mounted in the free-jet with a translating spark ignitor extended from above for ignition. Flame stability data were obtained and appear on the upper right plot where equivalence ratio is plotted versus the Dezubay flame stability parameter. The premixed stability limit determined experimentally by Dezubay is also shown. A Dezubay parameter value of about 1000 may be construed as a practical limit from this figure. An altitude versus Mach number chart appears in the lower right on which three trajectories are plotted along with the Dezubay limit of 1000 (for a 1-in. high flameholder). The 2-psia limit is also shown beyond which stable combustion is unlikely no matter what size flameholder is employed. It appears from this preliminary study that external burning is operable to low supersonic Mach numbers with lower altitude trajectories being stable over a wider range.

External Burning Drag-Reduction Experiment
Lewis PSL-4

Model Mounted in Mach-1.26 Free-Jet



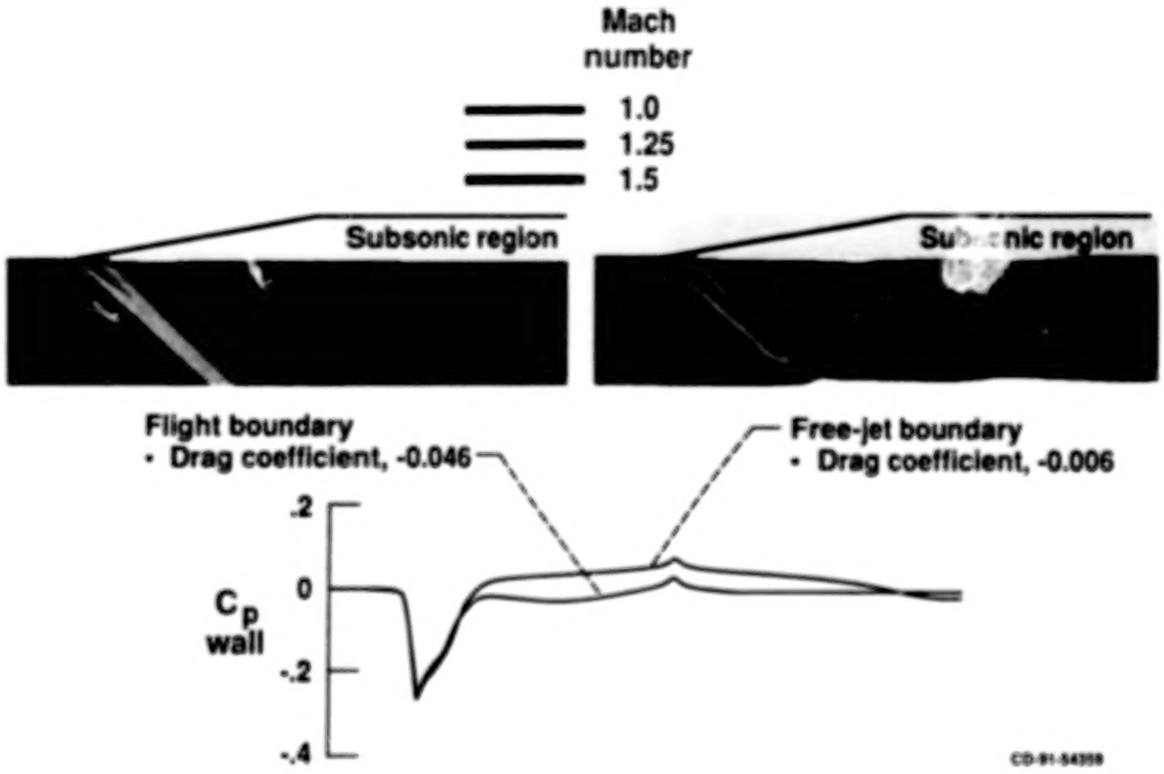


CD-91-54356

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The expansion ramp tests were intended to demonstrate drag reduction on a generic expansion ramp geometry, while providing calibration and verification information for analysis methods. No changes were made to the facility, and the expansion ramp models were mounted in the free-jet in much the same way as the spraybar models. The basic configuration, which is shown in the photograph, consisted of a 3- by 6-in. flat plate with a sharp leading edge, followed by an 11.2-in. long, 12° expansion ramp. Fuel was injected through a row of sonic orifices that were 1/2-in. upstream of the start of the expansion. A 1/8-in. high flameholder was employed to anchor the flame at the expansion. The model was tested over a range of fuel pressures and altitudes resulting in equivalence ratios from about 0.5 to 1.2. On the right is a plot of thrust coefficient (obtained by pressure-area integration) versus equivalence ratio which shows that performance is somewhat less than predicted by the control volume analysis. This performance decrement is the subject of current investigations and is attributed to poor combustion efficiency and to interactions of the free-jet boundary with the external burning plume.

Euler Analysis Used to Assess Free-Jet Boundary Interaction



A two-dimensional Euler analysis with heat addition (Euler + Q) was used to assess the magnitude of the free-jet interactions. Multiple wave reflections between the free-jet boundary and the external burning plume can be seen in the Mach number contour plots. The net result of this interaction is to lower the magnitude of the axial force over the entire range of equivalence ratios. As can be seen in the pressure coefficient plot for a low equivalence ratio case, the free-jet boundary interaction results in higher pressures and increases performance. The opposite is true for cases where thrust is generated.

NASP Jet–Exit Rig Further Extends Low–Speed Database

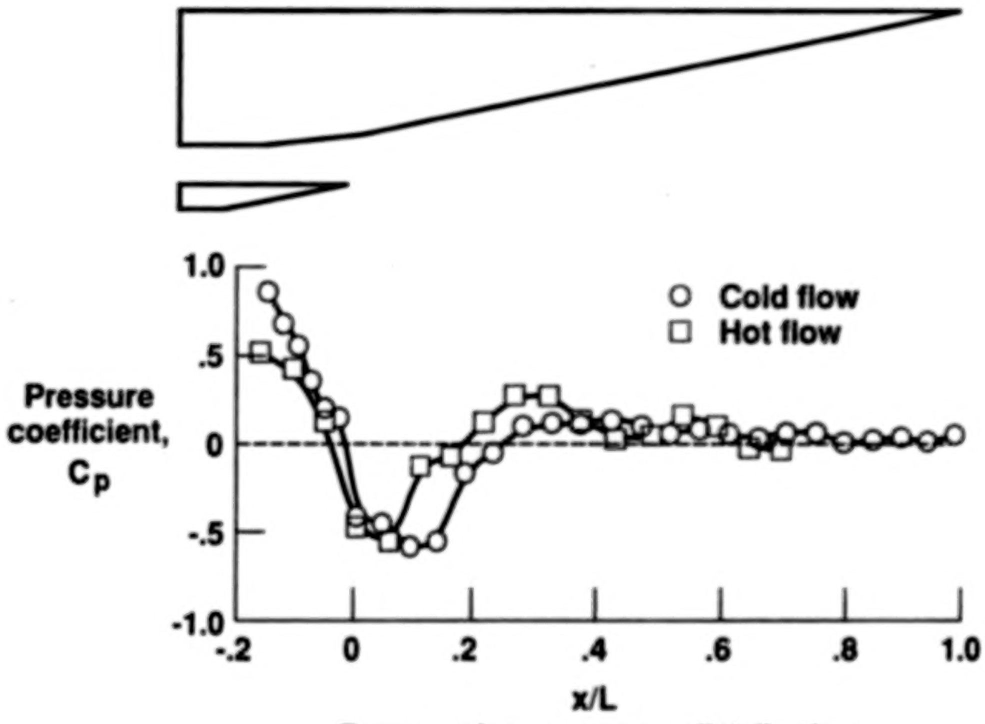
- **Hydrogen-air combustor simulates ramjet exhaust**
- **External burning capability**

CD-91-54360

This photograph shows the newly designed and fabricated jet exit rig installed in the Lewis 8- by 6-Foot Supersonic Wind Tunnel. This rig will be used to perform tests on nozzles of various geometries. It contains a hydrogen-air combustor, a six-component force balance, and an internal flow measurement system. The first use of this test rig will be in support of the NASP program to extend the low-speed nozzle/afterbody performance database with hot flow data. The hydrogen-air combustor can provide temperatures up to 4500 °R, but will be used to supply ramjet exhaust flow up to 3500 °R for this test. In addition, the use of external burning as a transonic drag reduction technique will be evaluated. Infrared flow imaging will be used to qualitatively evaluate the nozzle/afterbody and external burning plumes interaction with the tunnel flowfield. Heat transfer, temperature, and static pressure will be measured as well as force.

ORIGINAL PAGE IS
OF POOR QUALITY

Comparison of Hot and Cold Pressure Distributions



Ramp surface pressure distribution

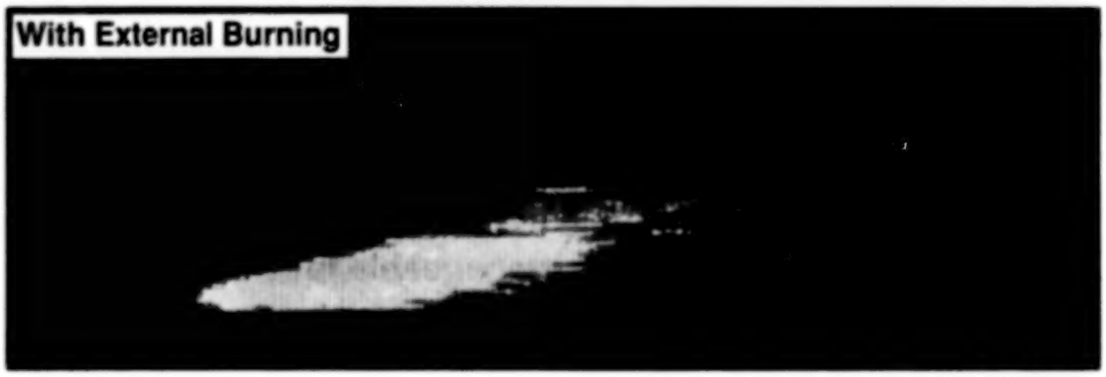
CD-91-54361

NASP nozzle testing to date at NASA's Langley and Lewis Research Centers has been conducted with cold air as the test medium. Questions naturally arise as to the scaling of this data to flight conditions, where hydrogen-air combustion products comprise the working fluid. Using the Jet-Exit Rig, the first NASP nozzle performance data using realistic combustion products is being obtained. Shown in this figure is a comparison between cold data taken at the Langley 16-Foot Transonic Wind Tunnel, and hot data taken recently in the Lewis 8- by 6-Foot Supersonic Wind Tunnel with the Jet-Exit Rig. The centerline pressure distributions shown look qualitatively similar, with some subtle differences near the cowl trailing edge. The implications of these subtle differences are currently being assessed.

Jet–Exit Rig External Burning Capability

Infrared Images

No External Burning



With External Burning

CD-91-54362

External burning as a means to reduce transonic nozzle drag is currently being investigated. Infrared images taken during a recent test show the Jet-Exit Rig in operation with and without external burning. The external burning hydrogen fuel is injected through a row of sonic orifices just upstream of the cowl trailing edge at pressures from 50 to 1000 psia. Ignition of the external burning stream is accomplished by the 3000 °R main exhaust flow.

ORIGINAL PAGE IS
OF POOR QUALITY

Summary

- NASP nozzle is highly overexpanded at transonic conditions
- Experimental and computational work is underway to
  - Characterize performance of candidate geometries
  - Reduce transonic drag to manageable levels
- Performance data with hydrogen combustor are being taken
- External burning shows promise as a drag-reduction technique

CD-91-54363

N91-20094

A COMPARISON OF CFD PREDICTIONS AND EXPERIMENTAL RESULTS FOR A MACH 5 INLET

Lois J. Weir
NASA Lewis Research Center
Cleveland, Ohio

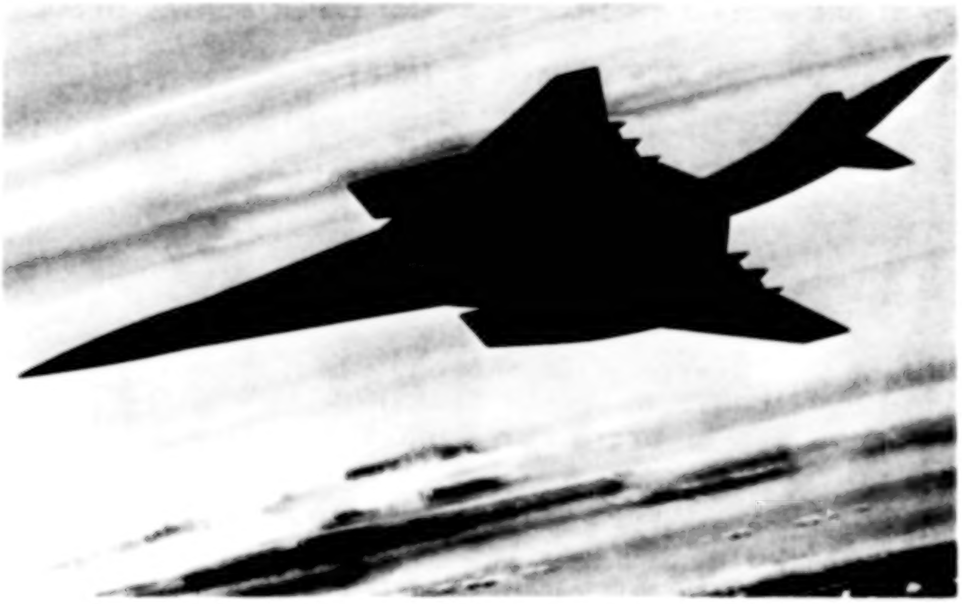
and

D.R. Reddy
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park, Ohio

Flow through a high-speed, nominally two-dimensional inlet is characterized by highly complex, three-dimensional flow phenomena including strong secondary flows and shock/boundary layer interactions. In this paper, experimental and analytical data are presented for a Mach 5 mixed-compression inlet that exhibits such three-dimensional flow characteristics. The two-dimensional inlet model, designed to provide performance information in the Mach 3 to 5 speed range for an "over-under" turbojet plus ramjet propulsion system, was also instrumented to provide computational fluid dynamics (CFD) code calibration and validation data, and, in particular, to provide some detailed data in regions where three-dimensional phenomena dominate the local flow field. Calculations of the inlet flow field presented include three-dimensional parabolized and full Navier-Stokes code analyses, with and without bleed. CFD analyses predicted flow migration up the inlet sidewalls due to glancing shock/boundary layer interactions, which, in turn, developed patterns of vortical flow in the inlet. This vortical flow was at least partially captured by the cowl, which set up large regions of low-energy flow in the corners underneath the cowl. As the vortical flow stream passed through the strong cowl shock, its vorticity appeared to be somewhat dissipated and the low-energy flow was compressed against the cowl.

Comparisons between analytical predictions and experimental results are presented for rakes located in regions of vortical flow, separation, or both in the corners between the cowl and the sidewalls. Both experimental and analytical results indicated that porous bleed upstream on the inlet sidewalls and in the corners of the cowl had little effect on the vortical flow entering the inlet; however, bleed removed farther downstream near the shoulder (after the rotational flow had passed through the cowl shock) appeared to remove the low-energy flow on the cowl and sidewalls rather effectively.

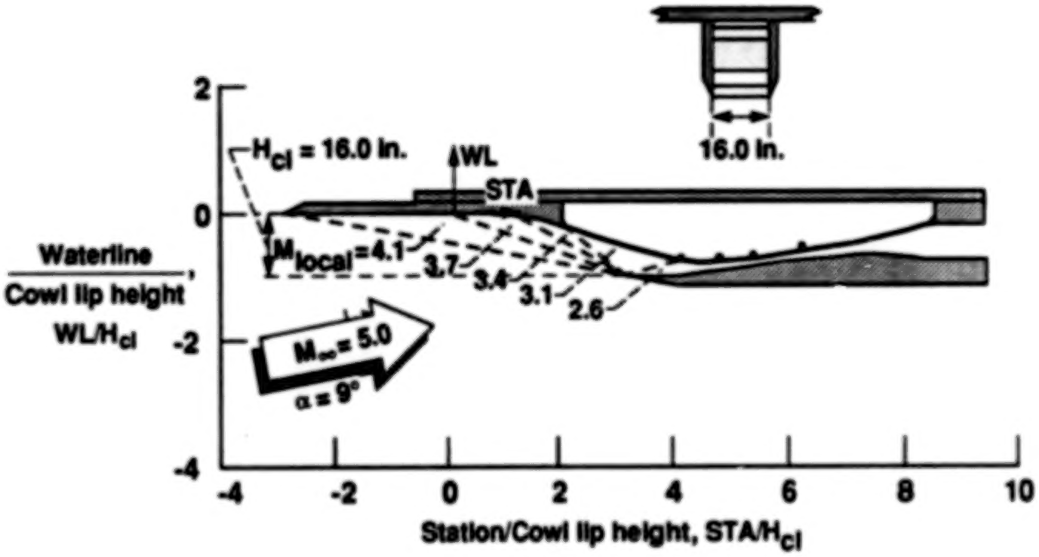
Artist's Concept of Mach 5 Cruise Aircraft



CD-81-53026

A joint research program between NASA Langley, NASA Lewis, and Lockheed (with Pratt & Whitney as a subcontractor) was begun in 1980 to investigate the critical flight technology issues in the Mach 3 to 5 speed range. The specific purpose of this study was to develop a concept for a Mach 5 penetrator aircraft, which is shown here. An over-under turbojet/ramjet propulsion system was chosen for this aircraft on the basis of size and weight considerations and ability to integrate with the airframe. Four of these propulsion systems were to be employed by the aircraft, two under each wing; and two-dimensional, dual-flow inlets were specified. A subsequent study resulted in the design of the lower (ramjet) portion of the inlet, which provides airflow in the Mach 3 to 5 speed range (ref. 1). A mixed-compression inlet model was designed for testing in the NASA Lewis 10- by 10-Foot Supersonic Wind Tunnel.

Inlet Design Contours


16.0 in.
H_{cl} = 16.0 in.
WL
STA
M_{local}=4.1
3.7
3.4
3.1
2.6
M_∞ = 5.0
α = 9°
Waterline / Cowl lip height, WL/H_{cl}
2
0
-2
-4
-4
-2
0
2
4
6
8
10
Station/Cowl lip height, STA/H_{cl}

CD-91-53827

The inlet design contours are presented here. The inlet was designed such that at Mach 5 cruise conditions, the incoming airflow was at an angle of attack α of 9° relative to the first ramp. This ramp decelerated the flow to Mach 4.1. Two fixed ramps each provided 5° more compression. The last ramp, which was variable for off-design operation, compressed the flow an extra 5° at Mach 5. The cowl lip provided 10° of compressive turning, and the cowl shock was canceled on the inlet shoulder at design conditions. Internal isentropic compression of 11° further decelerated the flow into the inlet throat. The variable ramp surfaces were hinged at the locations shown, and a slip joint was located near the last hinge to compensate for changes in length.

Inlet Model Installed in Wind Tunnel

CD-91-53030

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

A 1/3-scale model of the Mach 5 inlet was mounted on a trapezoidal plate in the NASA Lewis 10- by 10-Foot Supersonic Wind Tunnel. When set at a negative angle of attack, this plate accelerated the Mach 3.5 tunnel free-stream airflow to Mach 4.1, simulating airflow conditions on the first inlet ramp at Mach 5 cruise conditions. The accelerator plate was 100 in. wide, the inlet capture height (with the first ramp at 0° relative to free stream) and capture width were each 16 in., and the overall length of the model was approximately 20 ft. In addition to the variable ramp geometry, the model was equipped with remotely variable bleed exit plugs and main mass flow plug.

Inlet Model With Sidewall Removed

CD-91-53820

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The near sidewall was removed from the inlet to reveal the linkages that control the movement of the variable ramp surfaces. Also visible are several of the model's porous bleed regions, each of which was compartmented and controlled by a separate exit plug. Since sidewall bleed regions extended beyond the ramp position at design conditions, as shown, the sealed plates perpendicular to the ramp surface near the wall prevented recirculation of bleed flow into the cavity above the ramp. Also visible is some of the model instrumentation, which included surface static taps, fixed total pressure rakes, translating pressure probes, and dynamic pressure transducers.

Inlet Sidewall Bleed Exit Plugs

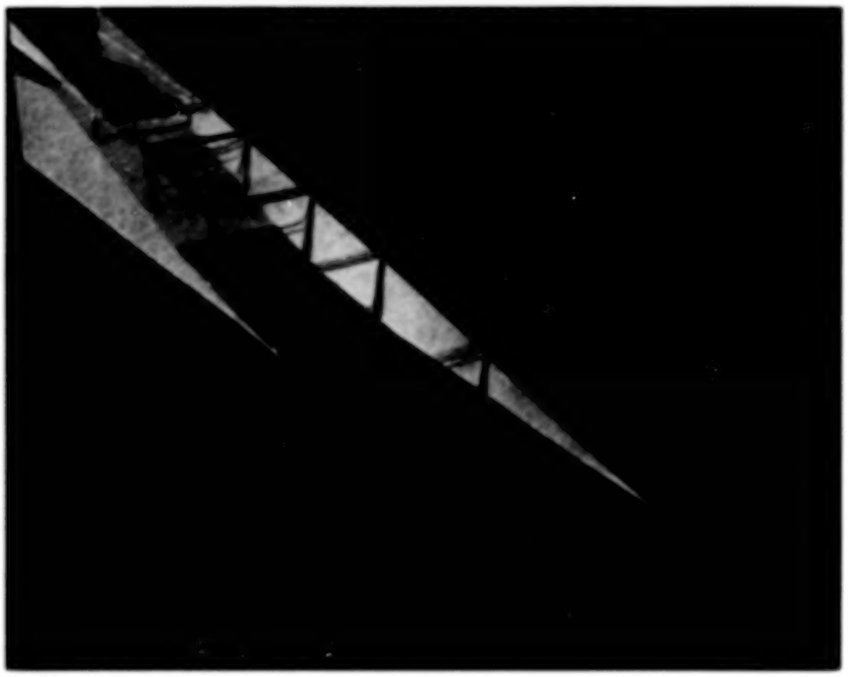
CD-81-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The photograph shows remotely variable bleed exit mass flow plugs on the inlet sidewalls. Airflow through 15 bleed regions (5 on the ramp, 2 on the cowl, and 4 on each sidewall) was controlled and measured by conical exit plugs like these.

PNS Analysis of Mach 5 Inlet

Mach Number Contours

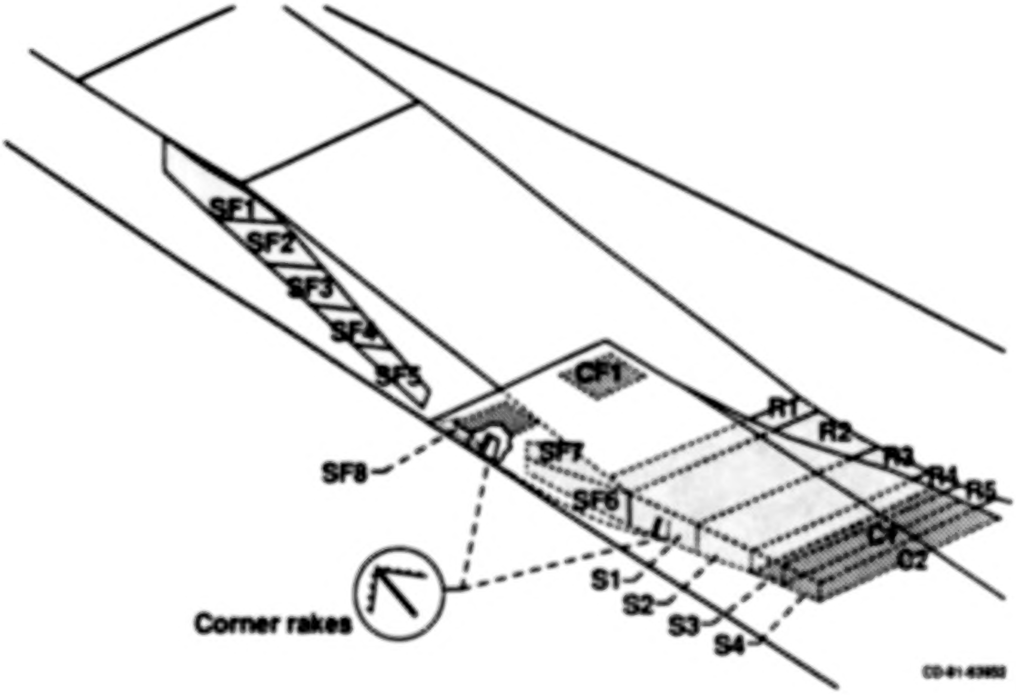


ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-[illegible]

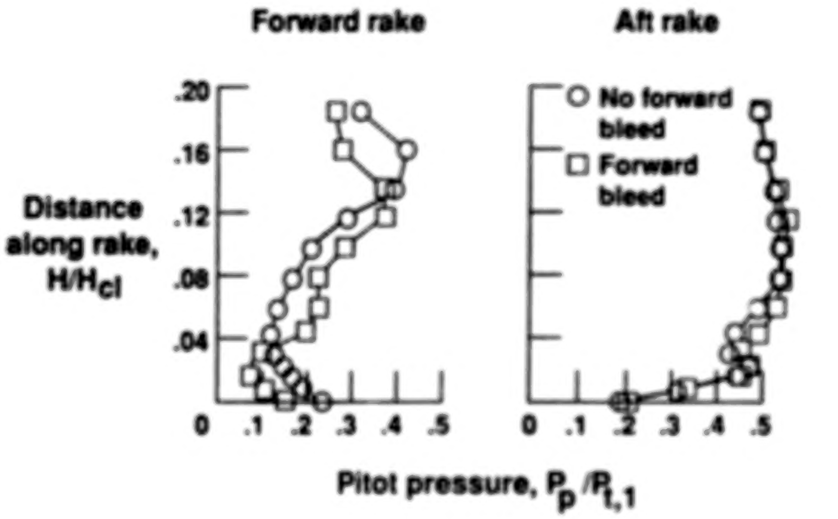
The Mach 5 inlet was designed two-dimensionally by using the method of characteristics, with the compression surfaces adjusted for displacement effects as predicted by viscous boundary layer codes. The first three-dimensional computational fluid dynamics (CFD) analysis was accomplished by Benson (ref. 2) using a parabolized Navier-Stokes (PNS) code, PEPSI-S (ref. 3). This analysis predicted a strong three-dimensional effect caused by the interaction of the ramp shock waves with the sidewall boundary layers. This glancing shock/boundary layer interaction phenomenon, which is described in more detail in reference 1, generated a vortical flow field near the sidewall boundary layer. This led, in turn, to boundary layer separations downstream of the cowl leading edge. This effect is illustrated in the figure, which shows Mach number contours at several inlet cross sections. The inlet, in this case, is inverted, with the cowl on the top. The farthest downstream cross section, aft of the cowl lip, indicates massive separations in the corner between the cowl and the sidewalls. Such separations, as indicated by this analysis, would likely lead to inlet unstart.

Inlet Bleed Regions and Corner Rake Locations



The figure indicates the locations of the various porous bleed regions on the inlet model. The original model design specified ramp bleed areas R1 to R5, cowl bleed areas C1 and C2, and sidewall bleed areas SF6 to SF8 and S1 to S4. Sidewall bleed regions SF1 to SF5 and cowl bleed region CF1 were added to the model design after the three-dimensional PNS analysis was completed. The purposes of this bleed were to diminish the three-dimensional flow phenomena caused by glancing shock/sidewall boundary layer interactions and to prevent separation on the cowl. Also shown are the locations of cowl corner rakes in the inlet. These rakes were oriented at an angle of 45° relative to the local cowl and sidewall surfaces. The farther upstream rake was located approximately 6 in. aft of the cowl leading edge (near the end of the first cowl corner bleed zone, $STA/H_{c1} = 3.73$), and the downstream rake was located 15 in. aft of the leading edge (slightly aft of the start of sidewall bleed region S1, $STA/H_{c1} = 4.28$). The placement of these rakes was chosen to coincide with the separation and vortical flow regions predicted by CFD.

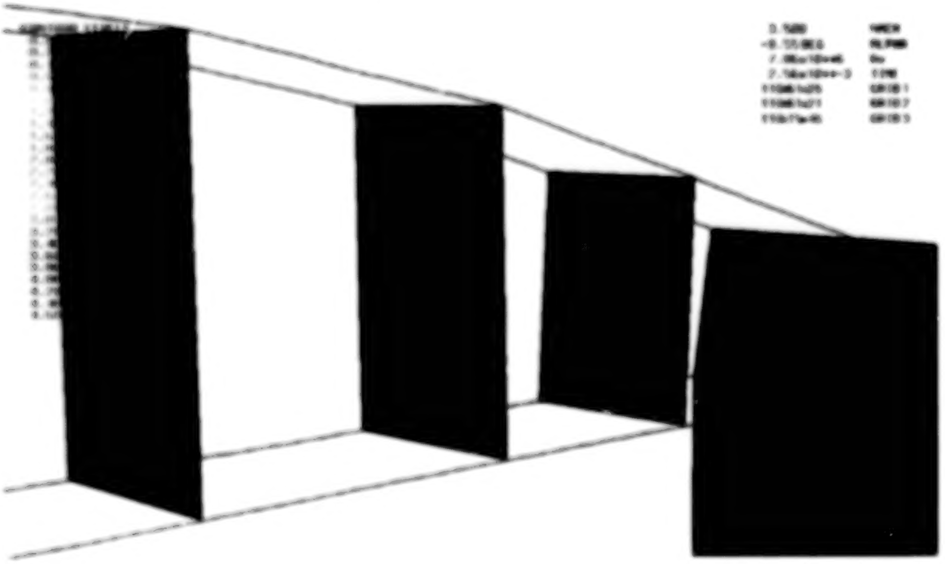
Cowl Corner Rake Pitot Pressure Profiles



CD-91-53853

Cowl corner rake pitot pressure profiles (nondimensionalized to tunnel total pressure $P_{t,0}$) are presented in the figure, data for the forward rake (STA/H_{cl} = 3.73) on the left and data for the aft rake (STA/H_{cl} = 4.28) on the right. For all data presented here the tunnel free-stream Mach number M_0 was 3.49, and the total pressure $P_{t,0}$ was 35.1 psia (this was also the total pressure on the first ramp $P_{t,1}$.) The angle of attack of the accelerator plate was -8.55°, providing a Mach number on the first ramp M_1 of 4.1, which simulated Mach 5 flight conditions. For all data presented here, bleed was removed from ramp bleed regions R1 to R5, cowl regions C1 and C2, and sidewall bleed regions S1 to S4. Bleed was also removed (square symbols) from sidewall bleed regions SF1 to SF8 and cowl corner bleed region CF1. These bleed regions were also closed off (circular symbols). The no-forward-bleed pitot pressure profile for the forward corner rake indicates large changes in pressure, with a region of medium pitot pressure near the corner that decreases rapidly, rises, and then decreases again near the end of the rake. The profile with forward bleed shows an additional small rise in pitot pressure after the initial falloff, and the pressure near the end of the rake fell off sooner. This seems to indicate that the rake was cutting across the zone of vortical flow in the cowl corner but that the flow was not the same with and without upstream bleed. Farther downstream (rake data shown on right), there is only a small "bump" in the profile near the corner, and bleed and no-bleed profiles are very similar. These data do not indicate the presence of vortical flow at this location. Either it has moved to an area where it cannot be detected by the rake, or it has been dissipated. In any case, upstream sidewall and cowl corner bleed seems to make little difference on the quality of the flow at this location.

Mach Number Contours
Rose and Perkins



CD-91-53694

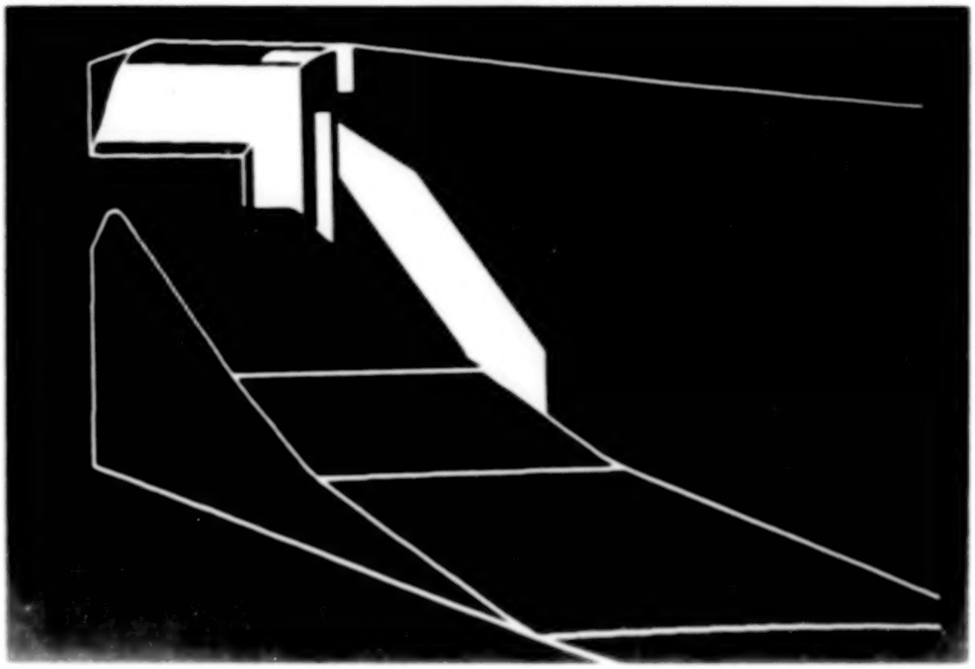
Several three-dimensional analyses of the inlet have been made recently using several codes. Two analyses are presented here, the first with a minimal amount of bleed (required to prevent the inlet from unstarting), and the second with large amounts of bleed removed from many bleed regions, simulating an experimental bleed configuration.

The first analysis shows the nature of the vortical flow region described earlier and its behavior when no bleed is present to control it. Rose and Perkins (ref. 5) calculated the flow field in the Mach 5 inlet by using the SCRAM3D code. This code uses the Kumar explicit, time-accurate implementation of MacCormack's algorithm to solve the full Navier-Stokes equations (ref. 6). The figure shows Mach number contours for half of the inlet cross section at several axial stations, with the near side being the inlet sidewall. The ramp is at the top of the figure, and the plane that will contain the cowl is at the bottom. The first contour plane is located at the junction of compression ramps 1 and 2. The second contour is at the junction of ramps 2 and 3. The third contour is at some station along ramp 3, and the last contour is at the cowl lip station. The thickening of the sidewall boundary layer can be seen in the upstream planes. In the second and third planes, the thickest part of the layer moves down the sidewall and begins to show vortical behavior. The cowl lip can be seen in the fourth plane as the thin dark band across the bottom of the inlet. The contour at the cowl lip shows that about one-third of the vortical flow is spilled over the cowl lip.

ORIGINAL PAGE IS
OF POOR QUALITY

Computational Bleed Zones

Reddy



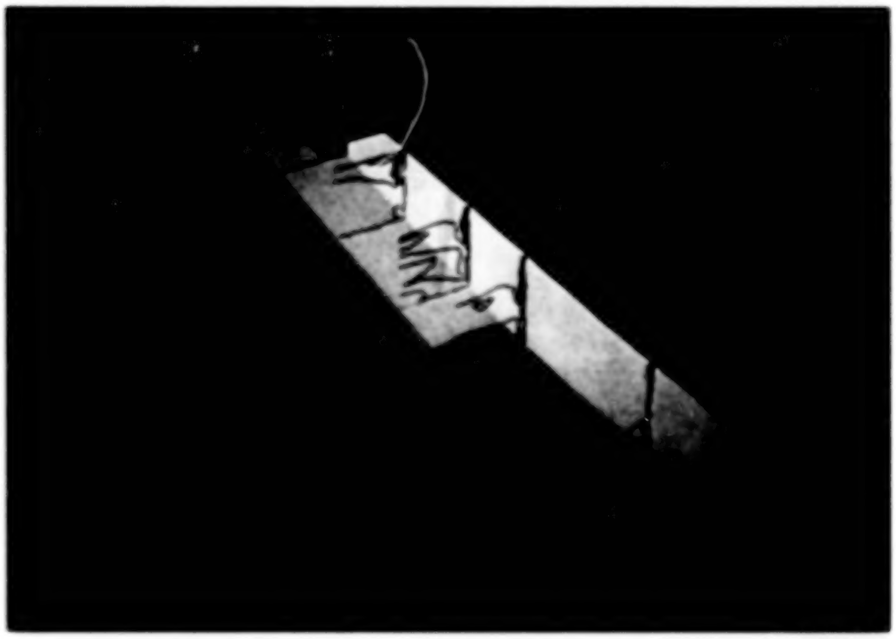
CD-91-53935

Another calculation of flow through this inlet was accomplished by Reddy, Benson, and Weir (ref. 7). A time-marching code, PARC3D (ref. 8), which solves the full three-dimensional Reynolds-averaged Navier-Stokes equations in strong conservation form with the Beam and Warming approximate factorization, was selected for this calculation. The turbulence model used in the code was the Baldwin-Lomax model. The calculation included a simulation of the bleed through the zones indicated here. In this figure, and in subsequent figures showing CFD predictions, the inlet orientation has been inverted, with the cowl on the top. The bleed was simulated in the computations by imposing a constant mass flow (measured in the experiment) through the bleed surfaces. A previous calculation (ref. 9) with no bleed showed a separation bubble near the throat region that grew slowly with increasing iterations, indicating that the inlet would unstart. Some bleed was, in fact, required to keep the inlet model from unstarting; thus, the calculation with bleed more accurately modeled the actual experimental conditions.

ORIGINAL PAGE IS
OF POOR QUALITY

Mach Number Contours

Reddy



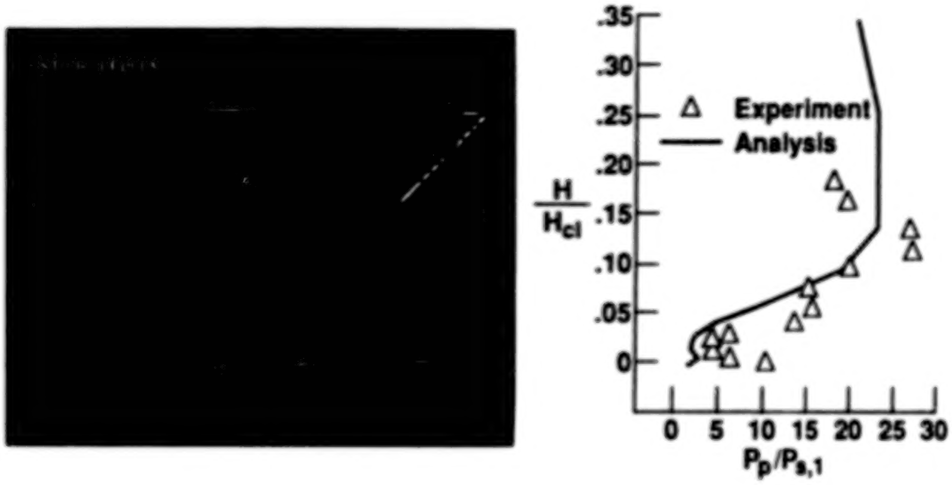
CD-91-53936

The figure shows Mach number contours at several inlet cross sections. Only half of the inlet flow field has been computed, since the inlet is symmetric in the transverse direction. Strong secondary flow is visible; it is set up by shock/boundary layer interaction and by migration of low-energy flow up the sidewall and then toward the inlet centerline as it is captured by the cowl and heads toward the throat. As the flow approaches the throat bleed regions, it appears that most of the low-energy flow on the cowl and sidewalls has been removed and that the Mach number profiles are basically clean.

ORIGINAL PAGE IS
OF POOR QUALITY

Comparison of Calculated and Experimental Upstream Corner Rake Data

Reddy

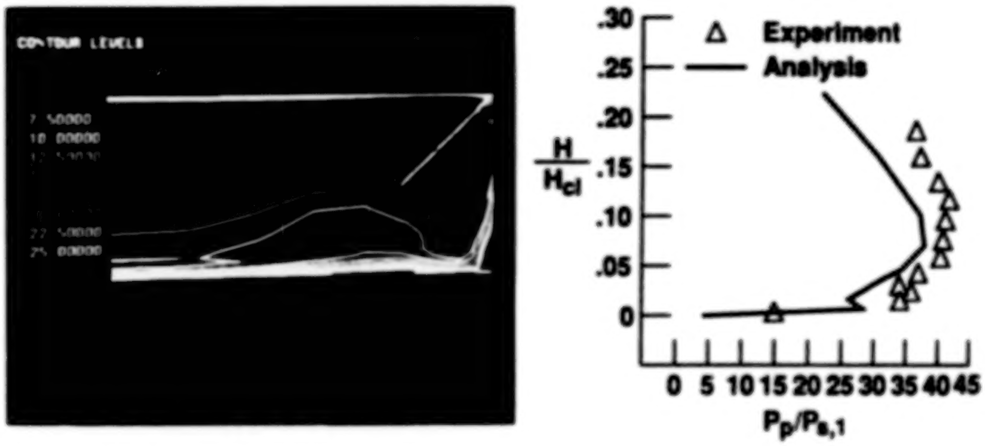


On the left side of the figure the orientation of the rake at STA/H_{c1} = 3.73 is shown relative to the pitot pressure ratio contours at a nearby station. On the right the calculated rake pitot pressure (nondimensionalized to plate static pressure, $P_{s,1}$) profile (solid curve) is compared with the experimental data (triangular symbols) with bleed. Although the calculation does not agree extremely closely with the data, it does indicate large pressure fluctuations across the region of low-energy flow in the cowl corner, indicating the presence of the vortical flow.

ORIGINAL PAGE IS
OF POOR QUALITY

Comparison of Calculated and Experimental Downstream Corner Rake Data

Reddy



CD-91-53838

A figure similar to the previous one is shown for the downstream corner rake (STA/H_{cl} = 4.28). In this case the analysis somewhat underpredicts the value of the pitot pressure, but displays a similar shaped curve. The pitot pressure contours indicate that the low-energy corner flow has been mostly removed, leaving fairly uniform flow in the duct at that station. This is most likely due to the effect of bleed on the sidewalls through regions SF6, SF7, and S1 (refer back to the figure showing inlet bleed areas), which has apparently removed much of the low-energy flow.

ORIGINAL PAGE IS
OF POOR QUALITY

CONCLUDING REMARKS

The research completed on the Mach 5 inlet at this time does not provide a complete understanding of the complex three-dimensional flow phenomena that are characteristic of rectangular inlets in this speed range. Further analysis and experimental data are required to determine the detailed flow physics that control the three-dimensional flow phenomena generated by shock/boundary layer interactions. However, some initial conclusions can be drawn from the data presently available. The analyses performed to date on the Mach 5 inlet agree that a vortical flow pattern is generated due to shock/boundary layer interactions. The data, particularly from the cowl corner rakes, tend to corroborate this. It is also generally agreed that bleed upstream on the sidewall, where the vortex is formed, has little effect on vortical flow. It appears that the nature of the vortical flow changes as it passes through the cowl shock and heads toward the throat, such that with bleed downstream the low-energy flow has been mostly removed and relatively uniform flow remains. Experimental data support this theory because the phenomena are no longer visible at the second corner rake station. Further analysis, data, or both are required to determine which bleed region or regions actually remove the low-energy flow.

In the future an identical test case (maximum bleed) will be run by using the SCRAM3D code to provide a direct comparison with the most recent PARC analysis. Other analytical efforts will involve modeling of alternative bleed configurations to determine the bleed locations required to remove the vortical flow. This effort will be supported by the second experiment phase, scheduled for August 1991. The primary objective of this phase will be to fully determine the performance of the inlet. However, additional data, including surface oil flow visualization, will be obtained where possible to help determine the size, extent, and behavior of the vortical flow.

REFERENCES

1. Perkins, E.W.; Rose, W.C.; and Horie, G.: Design of a Mach 5 Inlet System Model. NASA CR-3830, 1984.

2. Benson, T.J.: Three-Dimensional Viscous Calculation of Flow in a Mach 5.0 Hypersonic Inlet. AIAA Paper 86-1461, June 1986.

3. Anderson, B.H.: Three-Dimensional Viscous Design Methodology for Advanced Technology Aircraft Supersonic Inlet Systems. AIAA Paper 84-0194, Jan. 1984 (also NASA TM-83558).

4. Coltrin, R.E.: High-Speed Inlet Research Program and Supporting Analyses. Aeropropulsion '87, NASA CP-3049, 1987, pp. 469-486.

5. Rose, W.C.; and Perkins, E.W.: Innovative Boundary Layer Control Methods in High Speed Inlet Systems. NASA CR-185206, 1990.

6. Kumar, A.: Numerical Simulation of Flow Through Scramjet Inlets Using a Three-Dimensional Navier-Stokes Code. AIAA Paper 85-1664, July 1985.

7. Reddy, D.R.; Benson, T.J.; and Weir, L.J.: Comparison of 3-D Viscous Flow Computations of Mach 5 Inlet With Experimental Data. AIAA Paper 90-0600, Jan. 1990 (also NASA TM-102518).

8. Cooper, G.K.; Jordan, J.L.; and Phares, W.J.: Analysis Tool for Application to Ground Testing of Highly Underexpanded Nozzles. AIAA Paper 87-2015, June 1987.

9. Weir, L.J.; Reddy, D.R.; and Rupp, G.D.: Mach 5 Inlet CFD and Experimental Results. AIAA Paper 89-2355, July 1989 (also NASA TM-102317).

N91-20095

INTERNAL FLUID MECHANICS RESEARCH

Lonnie Reid and Louis A. Povinelli
NASA Lewis Research Center
Cleveland, Ohio

The Internal Fluid Mechanics Division is responsible for computational and experimental research on the internal aerothermodynamics of aeronautical and space propulsion systems. The research focuses on advancing the understanding of the relevant physics associated with improving the state of technology for propulsion system components. Research consists of the development of fast and accurate computational tools and models, the verification of these CFD tools and models through benchmark experiments, and their application to realistic propulsion system components. Advanced computational technologies are used to enhance, accelerate, and integrate computational and experimental research. The presentations to follow summarize ongoing work and indicate emphasis in three major research thrusts, namely, inlets, ducts, and nozzles; turbomachinery; and chemical reacting flows.

Goal

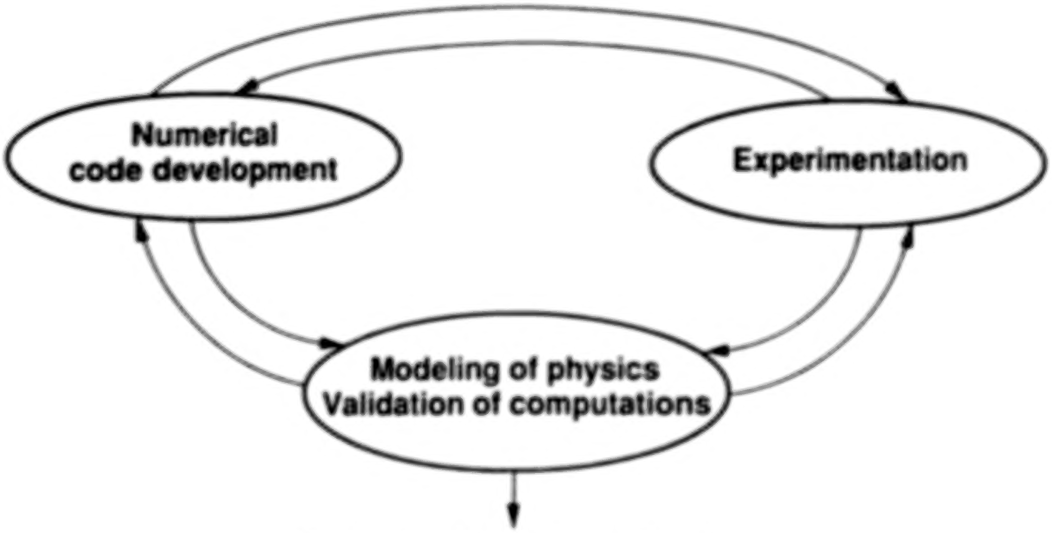
Bring internal computational fluid mechanics to a state of practical application for aerospace propulsion systems

Strategies

- **Provide state-of-the-art computational and experimental IFMD research programs.**
- **Combine advances in codes with emerging high-performance computing technologies to provide capabilities for interactive analysis and design optimization.**
- **Participate in cooperative research efforts to develop multidisciplinary analysis capabilities (e.g., NPSS).**
- **Develop closer working relationships with the propulsion industry and universities.**
- **Augment the research efforts of industry.**
- **Develop joint programs with industry.**

CD 91 53097

Closely Coupled Experimental and Computational Research



Numerical code development

Experimentation

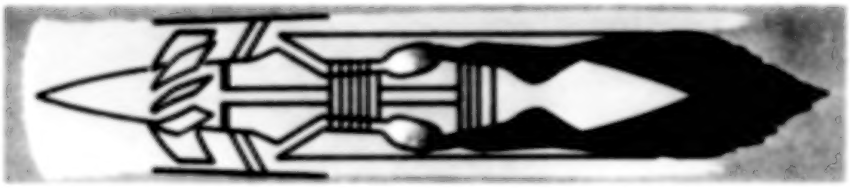
Modeling of physics
Validation of computations

Understanding of flow physics
Accurate predictive codes

CD-91-53998

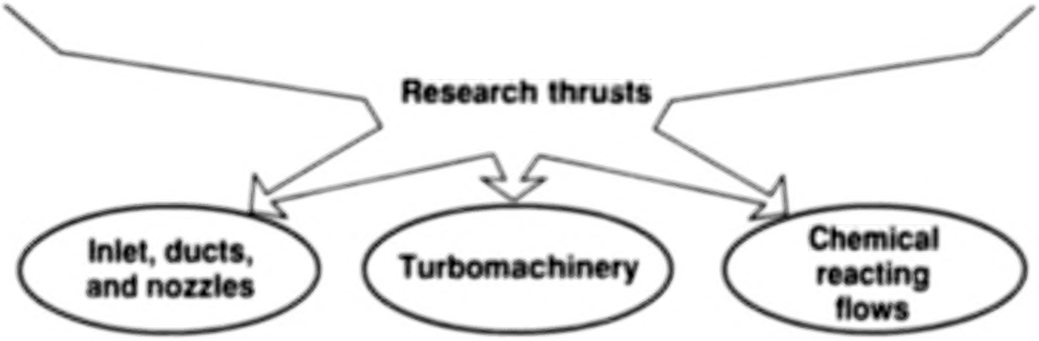
It is critical that numerical code development work and experimental work be closely coupled. The insights gained from experiments are represented by mathematical models that form the basis for code development. The resultant codes and models are then tested by comparing them with appropriate experiments in order to ensure their validity and determine their applicable range.

Research Thrusts



Highly 3-D flows   Unsteady flows   Turbulence/transition   Shear layers/mixing

Shock/boundary layer interaction   Multiphase flows   Chemical kinetics   Combustion



Research thrusts

Inlet, ducts, and nozzles

Turbomachinery

Chemical reacting flows

CD-91-53999

Propulsion systems are characterized by highly complex and dynamic internal flows. Many complex, three-dimensional flow phenomena are present, including unsteadiness, shocks, and chemical reactions. By focusing on specific portions of a propulsion system, it is often possible to identify the dominant phenomena that must be understood and modeled for obtaining accurate predictive capability. In the following presentations on internal fluid mechanics research, emphasis is placed on inlets, ducts, and nozzles, on turbomachinery, and on chemical reacting flows. These three research thrusts serve as a focus leading to greater understanding of the relevant physics and to improved computational fluid dynamics tools and models. This in turn will hasten advancements in propulsion system performance and capability.

N91-20096

INLETS, DUCTS, AND NOZZLES

James R. Scott and John M. Abbott
NASA Lewis Research Center
Cleveland, Ohio

The internal fluid mechanics research program in inlets, ducts, and nozzles is a balanced effort between the development and application of computational tools and the conduct of experimental research. The computational effort involves the development and validation of advanced computational fluid dynamics (CFD) codes through comparison with data, the modification of existing codes to extend their range and accuracy, and the application of codes to practical problems to demonstrate their value in design. The experimental research involves both simplified and realistic complex geometries and is used for developing flow physics understanding, for validating advanced numerical analysis codes, and for developing physical models of flow phenomena.

The inlet, duct, and nozzle research program is described according to three major classifications of flow phenomena: (1) highly three-dimensional flow fields, (2) shock and high-speed-mixing flow fields, and (3) shear flow control. Specific examples of current and future elements of the research program are described for each of these phenomena. In particular, the highly three-dimensional flow field phenomenon is highlighted by describing the experimental and computational research program in transition ducts having a round-to-rectangular area variation. In the case of shock and high-speed-mixing flow fields, both experimental and computational results are presented for the mixing of a high-speed stream injected into a second high-speed stream. For shear flow control, research in the use of aerodynamic excitation to enhance the jet mixing process is described. In addition, results that stem from using small tabs protruding into a nozzle exit flow stream to enhance mixing are also presented.

The presentation is concluded by describing a three-dimensional, unsteady viscous code development effort that will provide a well-documented, user-friendly flow solver for computing inlet, duct, and nozzle flow fields in the future.

Inlets, Ducts, and Nozzles Research Program

Flow phenomena	Drivers
Highly 3D flow fields	Unconventional engine location
	Thrust vectoring
	Maneuverability
Shock and high-speed-mixing flow fields	Higher flight speeds
Shear flow control	Shorter/lighter
	Enhanced mixing
	Acoustics

[illegible]

The internal fluid mechanics research program in inlets, ducts, and nozzles is described according to three types of fluid flow phenomena - highly three-dimensional flow fields, shock and high-speed-mixing flow fields, and shear flow control. The importance of each of these flow phenomena is a result of the drivers listed in the figure. For example, highly three-dimensional internal flow fields result from unconventional engine locations where twisting and turning inlets, ducts, and nozzles must be designed to deliver the airflow to and from the free stream. Aircraft thrust vectoring requirements quite often lead to the transitioning of nozzle cross-sectional geometries from round to rectangular with resultant three-dimensional flows. Aircraft maneuverability requirements can lead to strong three-dimensional flow fields entering the propulsion system inlet and ducting system. The push toward higher flight speeds, for both military and civilian applications, leads to the importance of research in shock and high-speed-mixing flows, particularly in terms of interactions (e.g., shock/boundary layer interactions). The desire to design inlets, ducts, and nozzles to be as short and light as possible points to the importance of shear flow control as a means of "stretching" the limits of the geometry while avoiding internal flow separations. Shear layer control in another sense (i.e., the use of aerodynamic excitation to control the formation and development of a mixing layer) offers the potential for enhancing the mixing process in external nozzle flows.

Inlets, Ducts, and Nozzles Research Program

Flow Phenomena

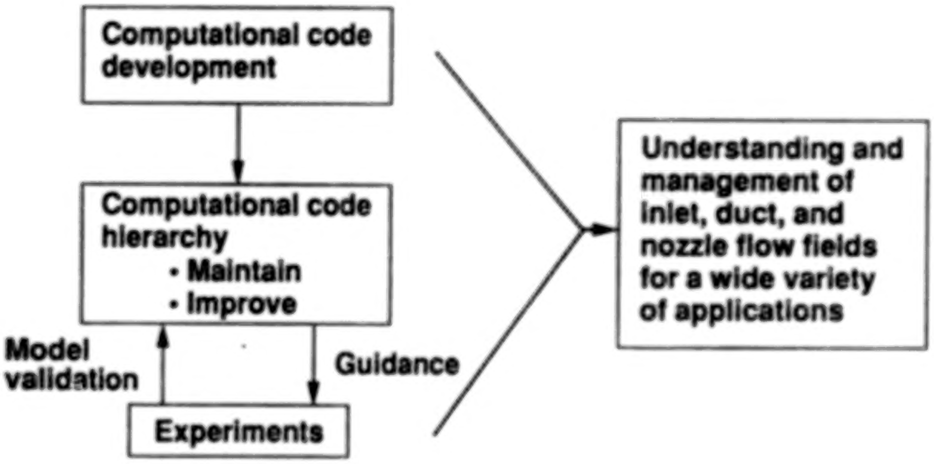
- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**
- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

CD-91-[illegible]

Specific elements of the inlet, duct, and nozzle research program are listed for each of the three flow field phenomena. The three elements highlighted in the figure will be expanded upon in the remainder of the presentation. Specifically, highly three-dimensional flow fields will be illustrated by describing the transition duct research program. The shock and high-speed-mixing flow field phenomena will be illustrated with a description of the high-speed-mixing research program. Shear layer control research will be illustrated with examples from the free-shear-layer control element of the program. The remaining three major elements of the overall program (i.e., offset ducts, shock/boundary layer interactions, and boundary layer control) will not be described in this presentation, although they are also key elements of the overall program.

Inlets, Ducts, and Nozzles Research Program

Approach



Computational code development

Computational code hierarchy
- Maintain
- Improve

Model validation

Guidance

Experiments

Understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications

CD-91-[illegible]

The approach to the research program in inlets, ducts, and nozzles is balanced between computational and experimental research. As shown in the figure, the ultimate goal of the program is to develop understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications. This is accomplished through an interactive process in which a strong computational effort of developing, maintaining, and improving CFD codes works hand in hand with an experimental effort that provides the data necessary for code validation and flow phenomena modeling. The codes within the hierarchy, in turn, provide guidance for conducting the experiments in terms of where measurements should be concentrated and what values of test parameters should be selected.

Inlets, Ducts, and Nozzles Research Program
Flow Phenomena

- Highly 3D flow fields
  - Transition ducts
  - Offset ducts

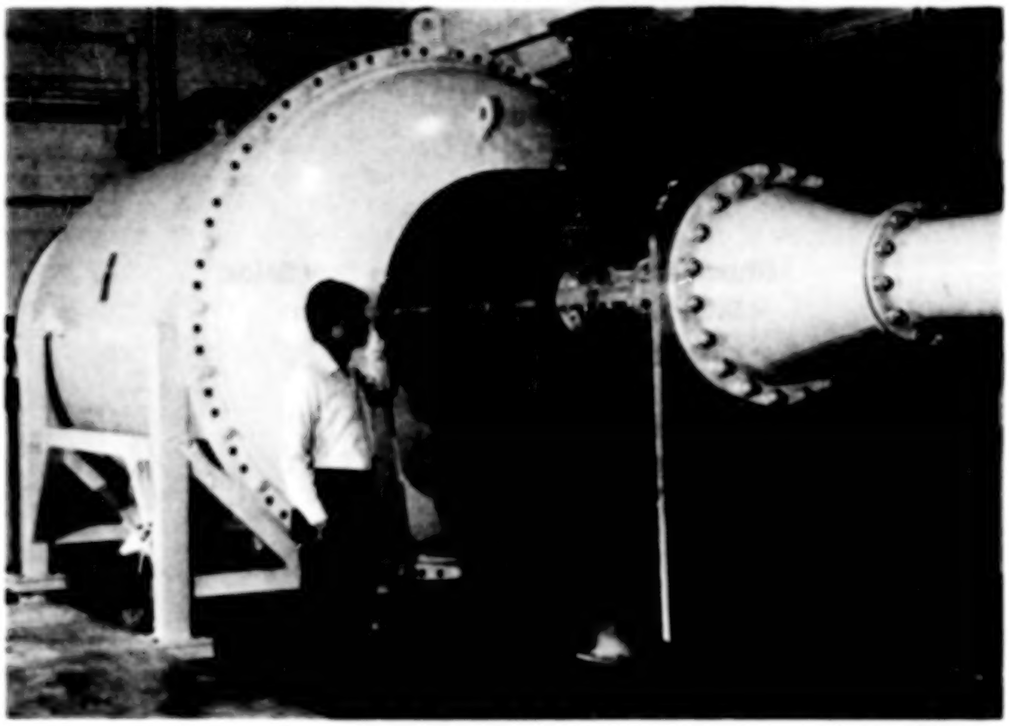
- Shock and high-speed-mixing flow fields
  - Shock/boundary layer interactions
  - Mixing

- Shear flow control
  - Boundary layers
  - Free shear layers

CD-91-54096

Highly three-dimensional flow field research will be illustrated by describing the transition duct research program.

Transition Duct Research



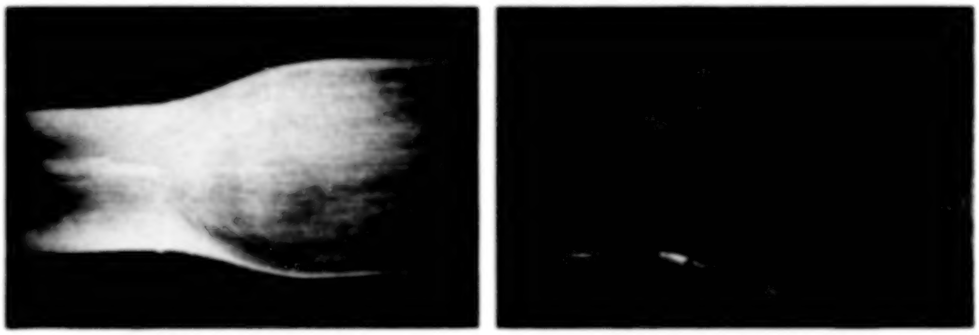
CD-91-5[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Transition ducts are characterized by a transition in cross-sectional shape from round to rectangular with downstream distance. Shown in the figure is a transition duct installed in the Internal Fluid Mechanics Test Facility. This particular duct geometry transitions from a round cross section to a rectangular cross section having an aspect ratio (width/height) of 3.0. The duct is three inlet diameters long and has an exit-to-inlet area ratio of 1.0. In the design of the test facility, special care was taken to condition the transition duct inflow properly to provide the desired levels of inflow turbulence, flow angularity, and boundary layer profile. In addition, a swirl generator was designed and built to provide a solid-body-rotation swirling flow into the transition duct for evaluation of how swirl affects duct performance. Swirl can become an important consideration for transitioning nozzle applications where residual swirl from the engine turbine may be convected into the transition duct. The facility is capable of being operated either with an atmospheric inlet (as shown here) or connected to a high-pressure supply system to provide higher Reynolds number test capability.

Transition Duct Flow Visualization Results

Axial Mach Number= 0.35



Surface oil streaks
(nonswirling inflow)

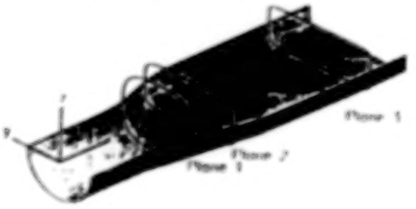
Surface oil streaks
(swirling inflow)

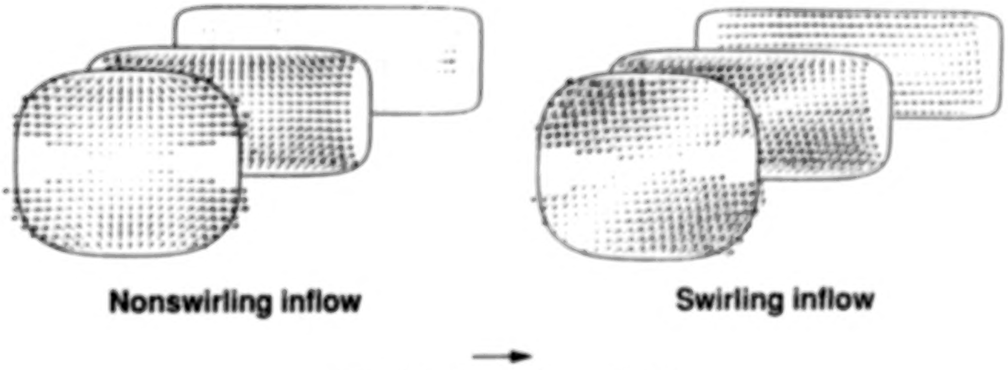
ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-[illegible]

Surface oil flow visualization results are shown in this figure for both the nonswirling and the swirling inflow cases with an inflow Mach number of 0.35. The flow is from left to right and indicates that for both cases the surface flow remained attached over the full length of the duct. The swirling inflow is clearly evident for the swirl case. The technique used to obtain these results consists of applying a uniform layer of oil containing a fluorescent dye to the surface of the duct and then operating the facility at the appropriate test condition for several minutes to let the oil flow and form streaks in the direction of the flow field near the surface. The duct is then disassembled and the resulting oil flow pattern is photographed using ultraviolet light for illumination.

Transition Duct Aerodynamic Results
Secondary Mach Number Vectors





Nonswirling inflow

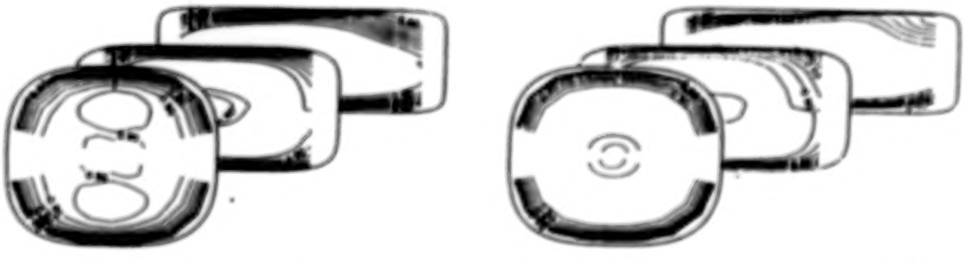
Swirling inflow

Inlet Mach number = 0.35

Experimental results are shown in this figure in terms of secondary Mach number vectors at three axial stations within the transition duct for both the nonswirling and the swirling cases. The duct inlet axial Mach number is 0.35, and for the swirl case the character of the swirl is close to that of solid-body rotation with an inlet swirl angle of about 16° at the duct wall. The "blank" regions within the duct represent areas where measurements could not be taken with the traversing probes. The values of Mach number were computed from measurements made with a calibrated five-hole probe. At each axial station, measurements were made at approximately 480 locations in the duct cross-sectional area. The results for the nonswirling inflow case show the development of a vortical flow structure at the duct exit plane as the flow has been influenced by the static pressure gradients associated with the duct area changes and curvatures in both the vertical and horizontal directions. Note also the side-to-side symmetry of the flow field. Top-to-bottom symmetry also existed in the duct, but the results shown here are from measurements made only in the lower half of the duct. The results shown for the upper half of the duct are rotated up from the lower half. The results for the swirling inflow case show the strong influence swirl can have on the flow field behavior. As expected, the upstream flow field clearly reflects the swirling inflow across the entire duct cross section and particularly striking is the disappearance of the downstream vortices that had developed for the nonswirling case. Clearly, swirling inflow can have a major effect on the flow field within a transition duct. Results like those shown in this figure were obtained for a range of centerline axial Mach number, and experiments were conducted on grant to obtain unsteady flow measurements for the exact same duct geometry.

Transition Duct—Comparison of Experimental and Computational Results

Total Pressure Contours
Inflow Mach Number = 0.35

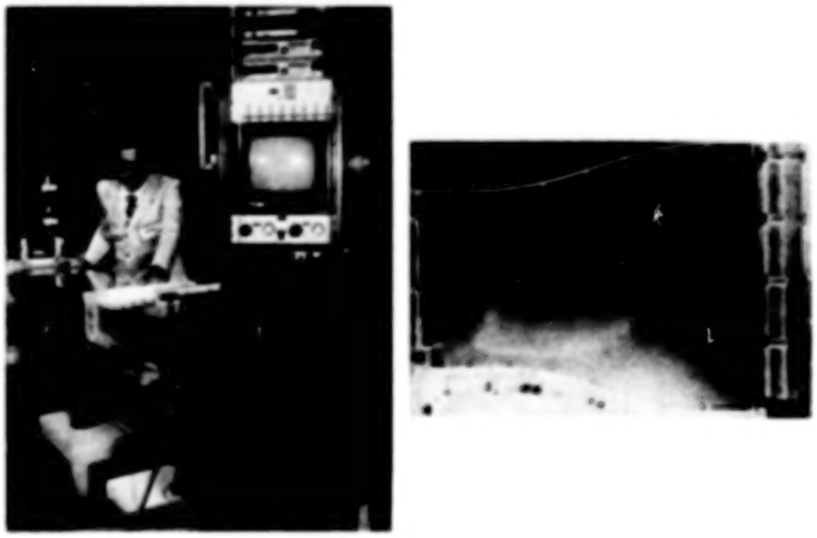


Experimental results **Computational results (PARC3D)**

CD-91-54099

A three-dimensional, steady Navier-Stokes calculation of the flow field within the very same transition duct geometry is compared in this figure with the experimental results. The results shown are contour plots of total pressure for the swirl case with an inflow Mach number of 0.35. The basic qualitative features of the flow are well resolved by the CFD predictions. At the first station, however, the experimental results show an oval-shaped contour line above and below the centerline that does not show up in the numerical results. It is believed that this feature of the flow may be an artifact of the upstream swirl generator, or that the upstream boundary condition used by the code is not adequately modeling the true incoming conditions of the flow. Overall, the agreement between the CFD predictions and the experimental results is quite good.

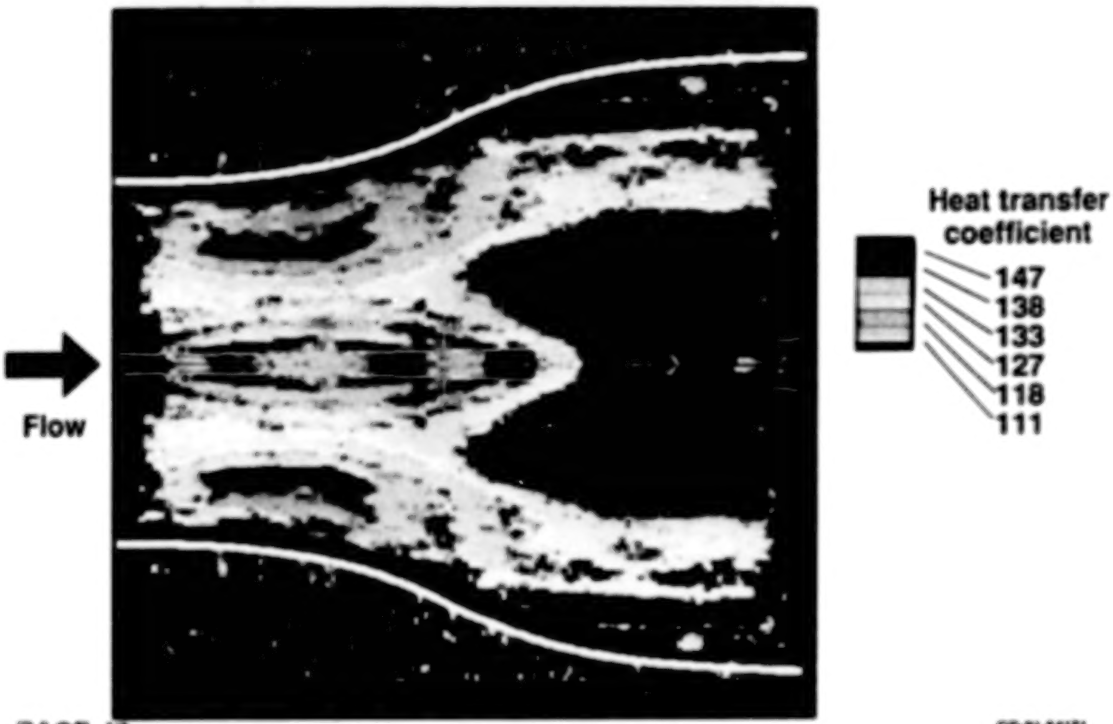
Transition Duct Heat Transfer Experiments



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-54100

In addition to aerodynamic experiments with the highly three-dimensional flow fields of transition ducts, heat transfer experiments are also being conducted. Heat transfer is of particular concern in applications where the three-dimensional flow may result in high-temperature flow streams (i.e., engine core flow) finding their way to the transition duct (nozzle) surfaces. The figure shows the Transition Duct Heat Transfer Research Facility and the transition duct. In order to measure the surface heat transfer characteristics of the duct, the surface was coated with liquid-crystal material. The duct was then heated to an elevated uniform temperature over its entire surface. A fast-acting valve was then opened and the desired flow rate of cooler room-temperature air was quickly established within the duct. The liquid-crystal material on the duct surface then responded to this changing surface temperature, and the corresponding changes in the transient behavior of the various temperature-sensitive color bands in the liquid-crystal material were recorded. From these recorded results the value of the local heat transfer coefficient at each location on the duct surface can be determined.



Transition Duct Heat Transfer Results

Inlet Mach Number = 0.14

ORIGINAL PAGE IS
OF POOR QUALITY

CD-91-54181

A heat transfer coefficient map for the transition duct, obtained by using the transient liquid-crystal technique, is shown in this figure. The results are for a duct inlet Mach number of 0.14. The various colors on the figure represent different values of surface heat transfer coefficient. The red area (shown on the centerline at the duct exit), where the airstream impinges on the duct surface, is the region with the highest heat transfer. The green areas (shown near the corner at the duct inlet and exit), where the airstream boundary layer thickens, are the regions with the lowest heat transfer.

Inlets, Ducts, and Nozzles Research Program
Flow Phenomena

- Highly 3D flow fields
  - Transition ducts
  - Offset ducts

- Shock and high-speed-mixing flow fields
  - Shock/boundary layer interactions
  - $\boxed{\text{- Mixing}}$

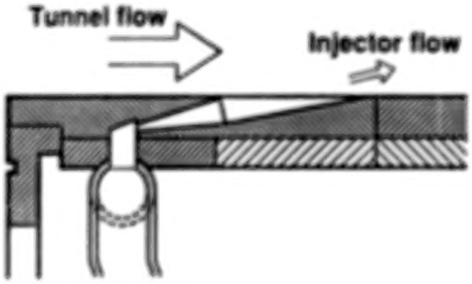
- Shear flow control
  - Boundary layers
  - Free shear layers

CD-91-54182

Shock and high-speed-mixing flow field research will be illustrated by describing the high-speed mixing research program.

High-Speed-Mixing Experiments

Single Flush Nozzle



Geometry

Schlieren

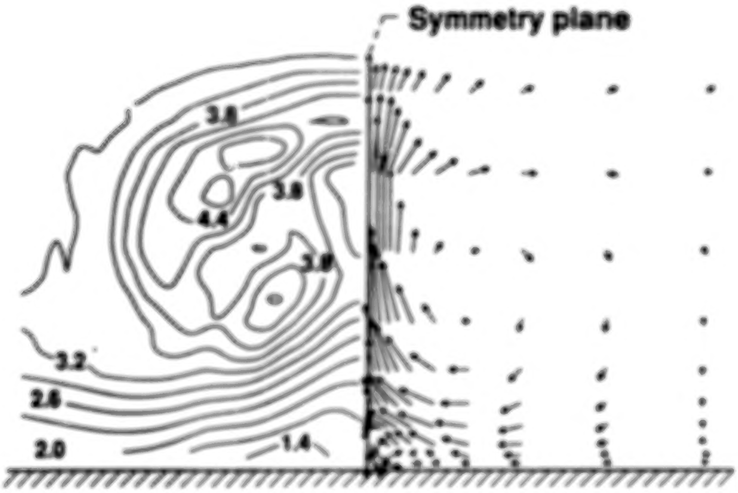
CD-91-54103

An experimental investigation is being conducted in the Lewis 1- by 1-Foot Supersonic Wind Tunnel to determine the effectiveness of various hypermixing nozzle concepts. The single flush nozzle shown in this figure was selected as a baseline model for investigating jet penetration and mixing over a range of operating conditions. Results from this nozzle configuration provide a comparison for results from more complicated multinozzle configurations. In addition, the simple geometry makes this model an ideal test case for code validation and calibration, inasmuch as complicated grid generation is avoided. The model consists of a circular underexpanded supersonic jet injected at a shallow (10°) angle into the boundary layer developing on the wind tunnel wall. The model was tested over free-stream Mach numbers from 1.6 to 3.0 with the nozzle-to-tunnel total pressure ratio varying from 6 to 30. The sample schlieren photograph illustrates some basic elements of the flow structure.

Single Flush Nozzle Results

Free-Stream Mach Number = 3.0; Jet Mach Number = 3.5;

$$\frac{\text{Downstream distance}}{\text{Jet exit diameter}} = 15.0$$



Experimental Mach number contours

Computational transverse velocity vectors (RPLUS 3D code)

Shown in this figure are both experimental and computational results for the single flush nozzle configuration. In both cases the free-stream Mach number is 3.0 and the jet exit Mach number is 3.5. The results are displayed 15 jet diameters downstream of the nozzle exit. The experimental results show a flow field that is characterized by the pressure-induced vortex pair associated with a jet in a crossflow. These vortices create the outward bulging of the contours away from the wall. In addition, the bulging of the Mach 1.4 contour near the wall, plus results from oil flow visualization photographs, hint that a second near-wall vortex pair may be present. The transverse velocity calculations shown on the right side of the figure indicate the presence of both the main vortices associated with the injection and the secondary vortices close to the wall. The agreement between the experimental and computational results appears to be quite good.

High-Speed-Mixing Experiments

Hypermixing Nozzle

Geometry

Schlieren

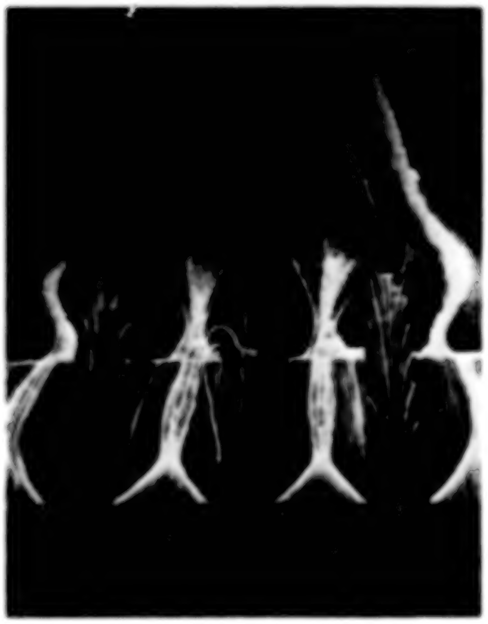
ORIGINAL PAGE IS
OF POOR QUALITY

[illegible]

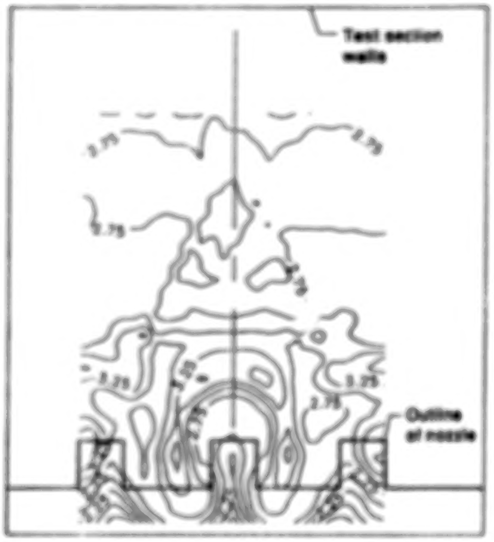
The second phase of the high-speed mixing program investigated the performance of multinozzle hypermixing configurations, one of which is illustrated in this figure along with a schlieren flow visualization photograph. The idea behind this type of nozzle concept is to utilize oblique shock wave interaction and streamwise vorticity to enhance penetration and mixing of the nozzle flow stream. The leading-edge ramp generates an oblique shock wave that then reflects from the opposing wall, crosses the jet plume, and favorably affects the jet mixing rates. In addition, the expansion troughs between the nozzles set up a pressure field that induces a strong vortex pair centered, at least initially, about the midspan of each individual nozzle. The desired effect of the vortex pair is to lift the jet up from the wall surface and thereby increase jet mixing. For the nozzle geometry shown here, the ramp angle is 10° and the jet injection angle is 15°. The free-stream Mach number ranged from 2 to 4 and the jet-to-free-stream total pressure ratio ranged from 1 to 12.

Hypermixing Nozzle Experimental Results

Free-Stream Mach Number = 3.0;
Jet Exit Mach Number = 3.0



Oil flow pattern



Mach number contours

$$\frac{\text{Downstream distance}}{\text{Nozzle height}} = 11.1$$

CD-91-54195

An oil flow visualization pattern and experimental results for the hypermixing nozzle configuration operating with a free-stream Mach number of 3.0 and a jet exit Mach number of 3.0 are shown in this figure. The oil flow pattern indicates very complex flow interactions; sidewall effects cause the near-wall flow to rapidly converge toward the wind tunnel centerline. With the exception of sagging in regions of heavy oil accumulation due to gravitational effects (the model was mounted on one of the wind tunnel vertical walls), spanwise symmetry was very good. The Mach number contours also indicate very good symmetry about the centerline of the geometry. The action of the vortex pairs (flow up) generated within the nozzle expansion troughs created the characteristic mushroom shape in the downstream contours.

Inlets, Ducts, and Nozzles Research Program

Flow Phenomena

- **Highly 3D flow fields**
  - **Transition ducts**
  - **Offset ducts**

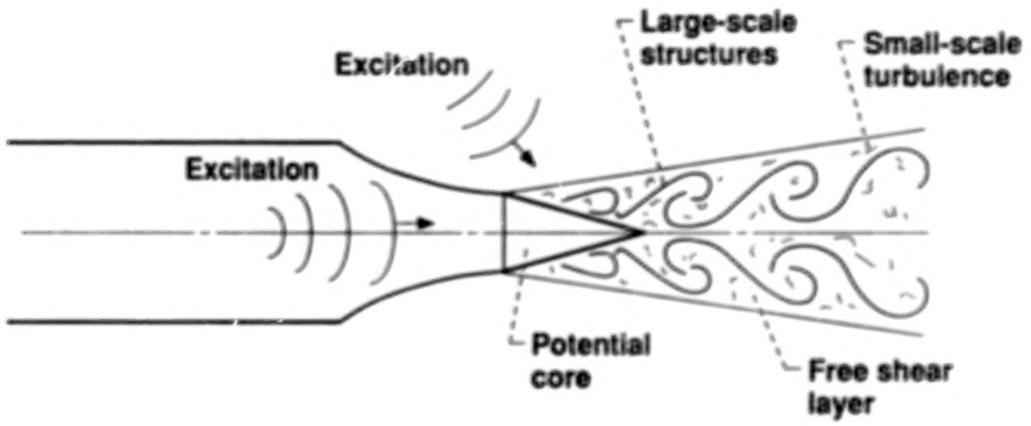
- **Shock and high-speed-mixing flow fields**
  - **Shock/boundary layer interactions**
  - **Mixing**

- **Shear flow control**
  - **Boundary layers**
  - **Free shear layers**

[illegible]

Shear flow control research will be illustrated by describing the free-shear-layer-control research program.

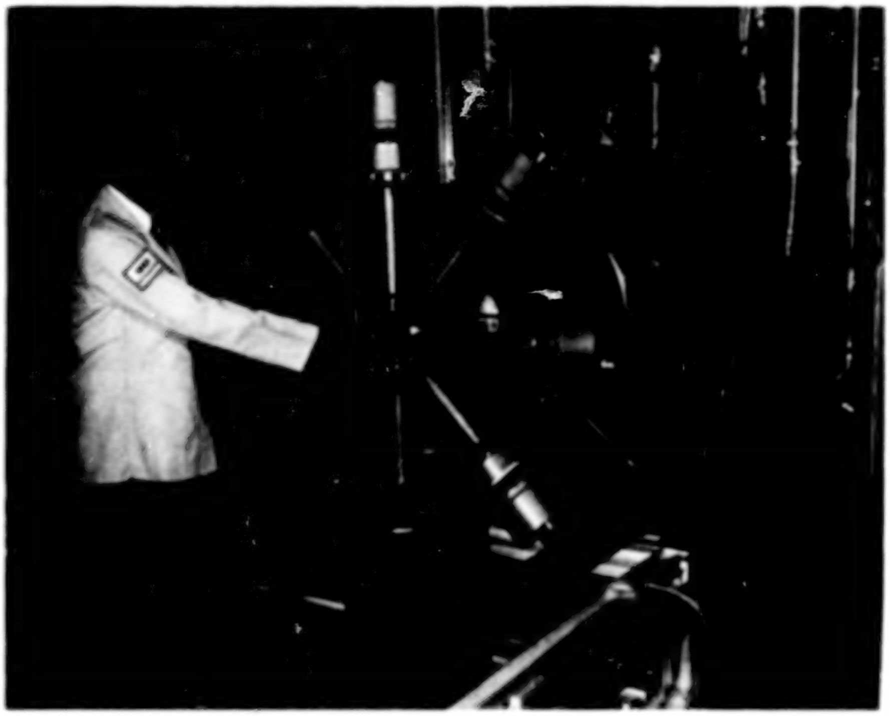
Enhanced Free-Shear-Layer Mixing


Excitation
Large-scale structures
Small-scale turbulence
Excitation
Potential core
Free shear layer

CD-91-54107

The mixing process between a jet and the surrounding airflow can be enhanced through use of aerodynamic excitation. The naturally occurring flow structure of the jet mixing process is illustrated in this figure. By applying excitation signals at the proper frequencies, with the proper modal structure, and with the appropriate amplitude and phase, the naturally occurring large-scale, unsteady structures within the mixing layer are regularized and enhanced and lead to a more rapid mixing process between the jet flow and the surrounding air.

Enhanced Free-Shear-Layer-Mixing Experiments



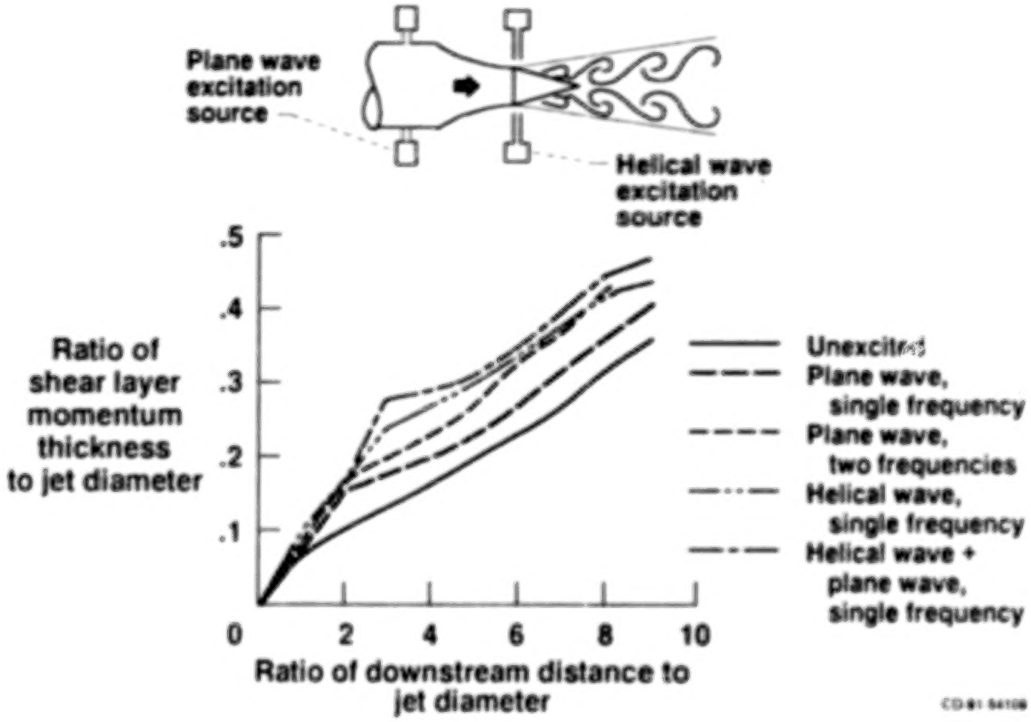
CD-81-54108

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This figure shows the test facility where experiments are being conducted to assess how aerodynamic excitation affects the free-shear-layer mixing process between a circular jet and the surrounding quiescent air. The facility has been carefully designed to provide a jet exit flow with a low turbulence level and with very low internal noise levels. The controlled aerodynamic excitation signals can be provided either internal to the plenum tank or externally. The external excitation system is clearly evident here and consists of the eight acoustical drivers arranged circumferentially around the jet exit plane. These external excitation devices are used to provide helical mode excitation; the internal excitation devices are used to provide plane wave excitation. Through proper control of frequency, amplitude, phase, and modal structure, any number of aerodynamic excitation combinations can be applied to the jet free shear layer.

Enhanced Free-Shear-Layer Mixing—Experimental Results

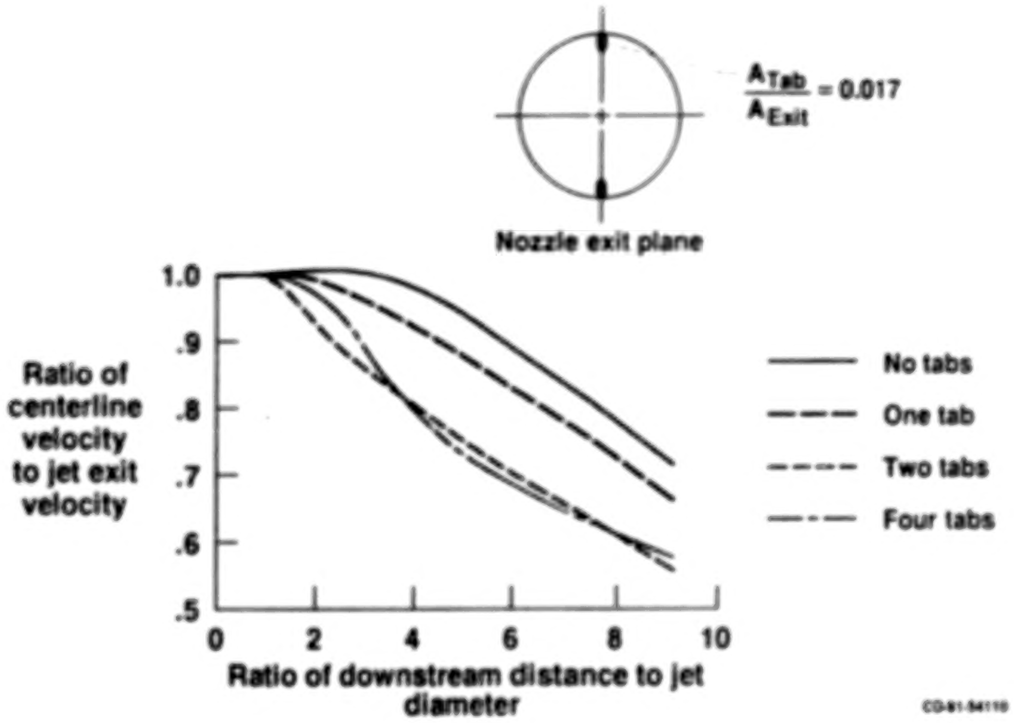
Jet Exit Mach Number = 0.2; Jet Turbulence = 0.15 Percent



The favorable influence of aerodynamic excitation on the mixing of a circular jet free shear layer is shown in this figure. The results are for various types of aerodynamic excitation in terms of the momentum thickness of the shear layer versus the downstream distance. The jet exit Mach number is 0.2 and the turbulence level within the jet at the exit is 0.15 percent. The lower curve on the figure represents the downstream growth of the jet shear layer for the natural, unexcited jet. The other four curves show the progressive effect on shear layer growth of (1) plane wave excitation applied at the preferred jet excitation frequency, (2) plane wave excitation applied at both the preferred jet excitation frequency and at half that frequency (to encourage large-scale structure "pairing"), (3) helical wave excitation applied at the preferred jet excitation frequency, and (4) both helical wave and plane wave excitation applied at the preferred frequency. The results indicate the significant influence that aerodynamic excitation can have on free-shear-layer mixing.

Enhanced Free-Shear-Layer Mixing—Effect of Tabs

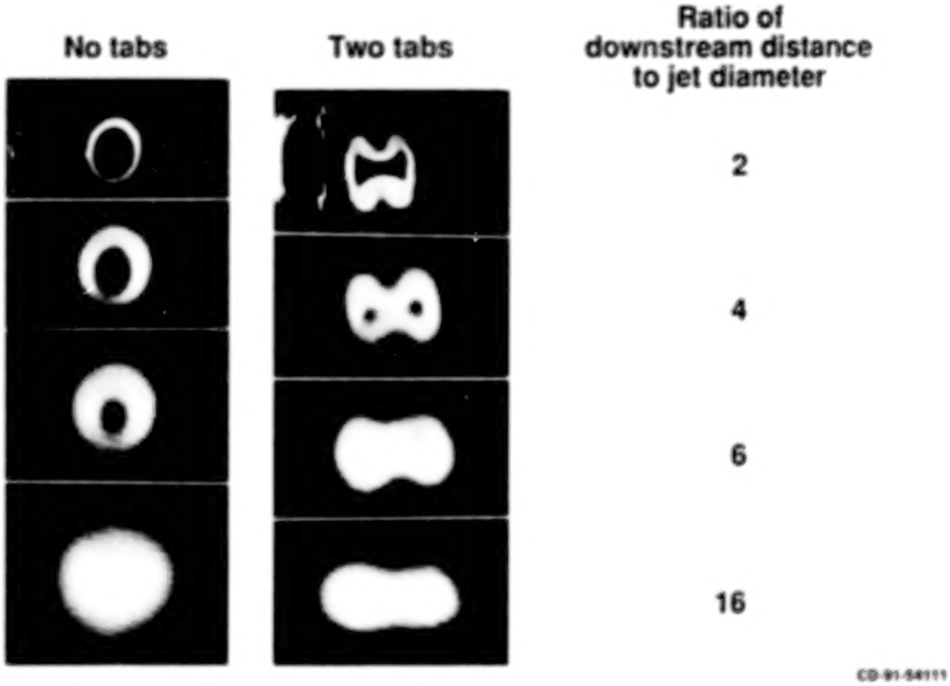
Jet Exit Mach Number = 0.34



In addition to investigating the effects of aerodynamic excitation on the mixing process of a free shear layer, the effect of tabs, or small protrusions in the flow at the exit plane of a nozzle, has also been investigated as part of the shear-flow-control research program. As indicated in the figure, these tabs are quite small, blocking only 1.7 percent of the exit flow area (per tab). How the tabs influence jet mixing is shown in this case in terms of the decay of the jet centerline velocity with downstream distance for a jet exit Mach number of 0.34. The top curve shows the natural decay of the jet centerline velocity for the jet with no tabs. The other three curves show the effect of adding one, two, and then four tabs at the nozzle exit plane. Note that significant increases in mixing (decreases in centerline velocity) occur with the introduction of one and then two tabs. The four-tab case is not too different from the two-tab case. These investigations were conducted over the Mach number range 0.34 to 1.81. Similar enhancements to jet mixing were observed at all jet exit Mach numbers.

Enhanced Free-Shear-Layer Mixing—Effect of Tabs

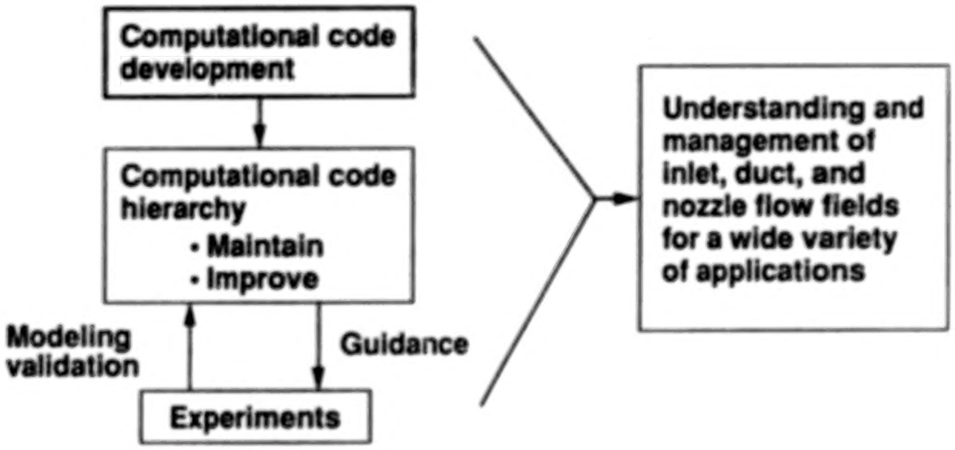
Jet Exit Mach Number = 1.63



The flow visualization results shown in this figure illustrate the changes that occur in the shape of the jet mixing layer when tabs are introduced into the nozzle exit plane. The jet exit Mach number was 1.63. Results are shown for the no-tab and the two-tab cases at four downstream locations. The images were obtained by passing a sheet of laser light through the jet, perpendicular to the jet centerline. Because of the high jet Mach number the static temperature of the jet flow was far enough below the room air dewpoint that the moisture in the room air, which had mixed in with the jet air in the shear layer, condensed and formed the images shown. As expected, for the no-tab case the mixing layer remained circular in cross section with the jet core vanishing just downstream of six jet diameters. The two-tab case, however, essentially split the jet core into two jets and this effect persisted until at least 16 diameters downstream. The jet clearly spread more in the plane perpendicular to the line containing the two tabs.

Inlets, Ducts, and Nozzles Research Program

Approach



Computational code development

Computational code hierarchy
- Maintain
- Improve

Modeling validation

Guidance

Experiments

Understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications

CD-91-54112

Thus far, this description of the internal fluid mechanics research program in inlets, ducts, and nozzles has focused on showing the results of experiments and the application of computer codes to several flow phenomena of interest. Returning to this earlier figure, another element of the program involves the development of new computer codes, which are added, as necessary, to the existing hierarchy of codes, to be used for the stated objective of improving the understanding and management of inlet, duct, and nozzle flow fields for a wide variety of applications. By way of closing this presentation, one such code development effort, currently under way, will be described.

PROTEUS Code Development

- **Objective**
  - **Develop user-oriented, easily modified, well-documented, two- and three-dimensional, unsteady Navier-Stokes codes for aerospace propulsion applications**
- **Approach**
  - **Use well-proven, state-of-the-art algorithms**
  - **Use a consistent, modular code structure**
  - **Emphasize documentation, code readability, and validation**

CD-91-54113

An effort is currently under way to develop two- and three-dimensional unsteady Navier-Stokes codes, called PROTEUS, for aerospace propulsion applications that include inlets, ducts, and nozzles. The emphasis in the PROTEUS effort is not algorithm development or research on numerical methods but rather the development of the code itself. The objective is to develop codes that are use oriented, easily modified, and well documented. Well-proven, state-of-the-art solution algorithms are being used. Code readability, documentation (both internal and external), and validation are being emphasized.

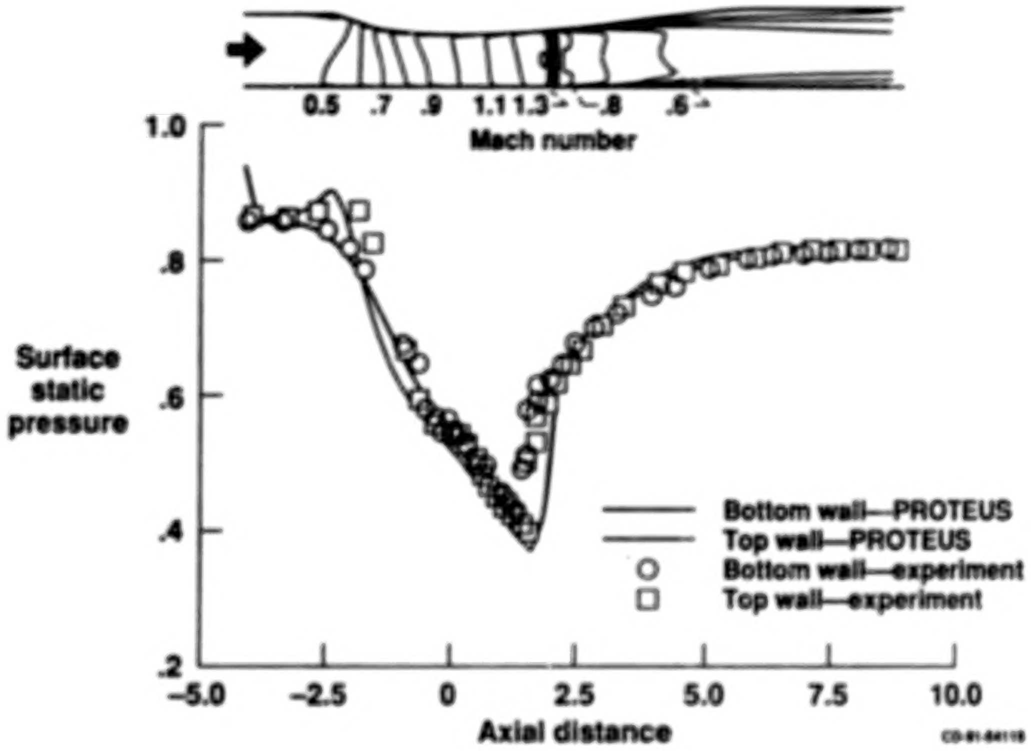
PROTEUS Validation Cases

- Flow over a flat plate
  - Blasius solution for laminar flow
  - Laminar flow with confined separation
  - Turbulent flow with or without heat transfer
- Exact Navier-Stokes solutions
- Driven cavity
- Development of laminar and turbulent flow in a channel and pipe
- Steady and unsteady turbulent flow in a transonic diffuser

CD-91-54114

An extensive series of validation cases have been run, primarily using the two-dimensional planar/axisymmetric version of PROTEUS. Some of the major validation cases are listed here and include comparing PROTEUS results with exact Navier-Stokes solutions, with experimental results, and with results from other numerical calculations.

Transonic Diffuser Validation



A typical PROTEUS validation case is shown in this figure. The geometry consists of a two-dimensional transonic diffuser operating at a maximum Mach number of about 1.3 prior to shocking down to subsonic conditions. PROTEUS results are compared with experimental data taken on both the top and bottom surfaces of the diffuser. The agreement between the PROTEUS results and the experimental data is quite good over the full length of the diffuser.

PROTEUS Code Status

- Two-dimensional version released in September 1989
- Three-dimensional version release scheduled for September 1991
- Users workshop to be held in spring of 1992

CD-91-54118

A two-dimensional version of the PROTEUS code was released in September 1989 along with a three-volume set of documentation. The three-dimensional version of the code is scheduled for release in September 1991. A PROTEUS users workshop will be held in the spring of 1992.

N91-20097

TURBOMACHINERY

Anthony J. Strazisar, Raymond E. Gaugler,
John J. Adamczyk, and Dale J. Arpasi
NASA Lewis Research Center
Cleveland, Ohio

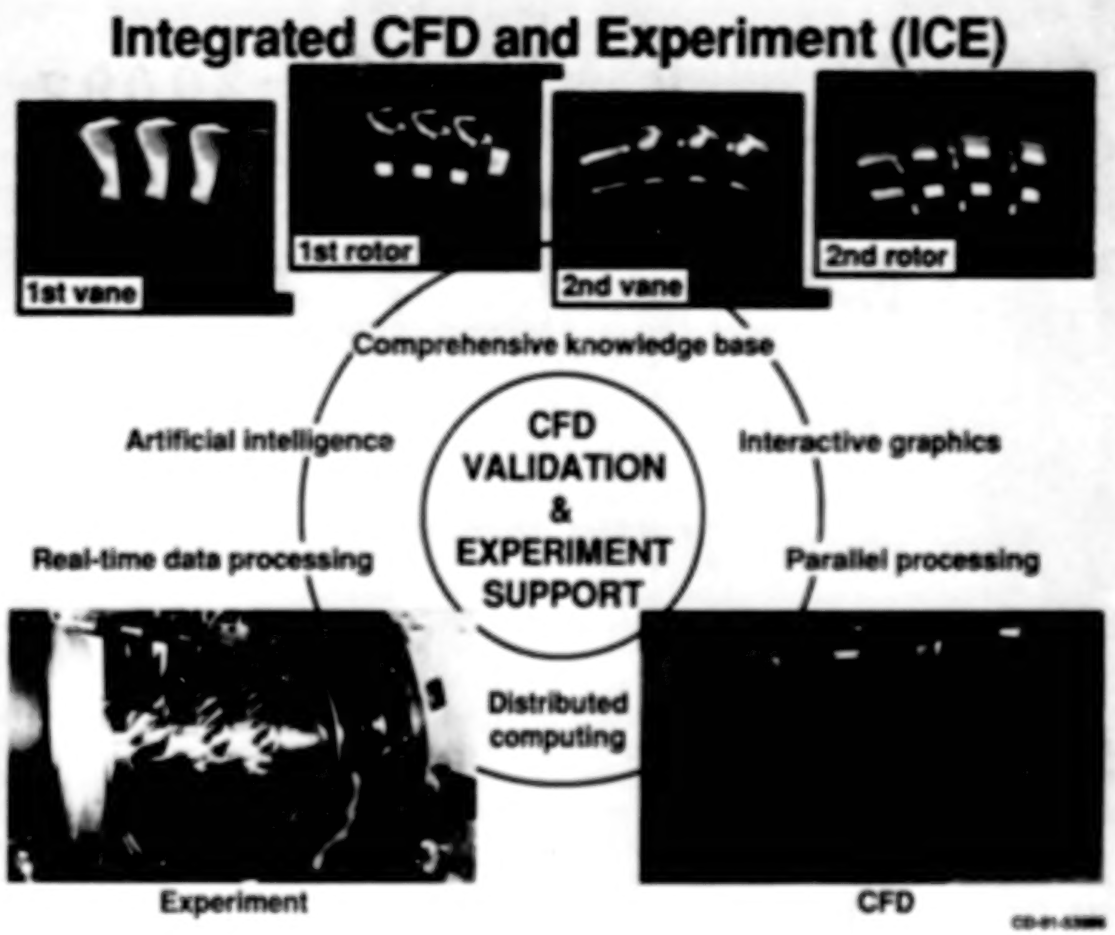
and

Isaac Lopez
Propulsion Directorate
U.S. Army Aviation Systems Command
Lewis Research Center
Cleveland, Ohio

We are responsible for developing the computational capabilities and furthering the understanding of flow physics phenomena fundamental to the advancement of turbomachinery technology. Our research involves the development, assessment, and application of computational fluid dynamics (CFD) tools for turbomachinery flows and the acquisition and analysis of experimental measurements of flows in turbomachinery. The measurements involve both simplified and realistic, complex geometries, and are used both for the validation of advanced numerical flow analysis codes and for the development of physical models for flow phenomena. They are obtained in both large scale, low speed facilities and full scale, high speed, transonic facilities. The computational emphasis involves the development of advanced computer codes, their assessment by comparison with quality data, modification of codes to extend range and accuracy, and application of codes to practical problems to demonstrate their value in design and in providing detailed description of flow field phenomena. The capability of the CFD codes to accurately calculate losses and heat transfer in 3D viscous steady and unsteady flows is emphasized. Through interactions with other Lewis organizations, these capabilities are applied toward the solution of technical problems important to the mission of the NASA.

Our research involves experiments using advanced instrumentation systems such as laser and hot wire anemometry in order to resolve spatial and temporal flow field variations. Major facilities include a full scale axial flow compressor, a large, low speed centrifugal compressor, a low speed multistage axial research compressor, and a full annular turbine stator cascade.

We synergistically apply experimental and computational efforts to maximize our understanding of the complex flows existing in turbomachinery passages. We apply 3D viscous steady state computational codes to both axial and radial turbomachinery with emphasis on the prediction of the detailed blade row performance and local heat transfer coefficients. We apply 2D and 3D unsteady codes to evaluate the impact of flow field deterministic unsteadiness on stage performance.



Integrated CFD and Experiment (ICE)

1st vane
1st rotor
2nd vane
2nd rotor
Comprehensive knowledge base
Artificial intelligence
Interactive graphics
CFD VALIDATION & EXPERIMENT SUPPORT
Real-time data processing
Parallel processing
Distributed computing
Experiment
CFD

The ICE Program was created to meet the needs of research programs such as the Multistage Compressor Flow Physics Program which feature tightly coupled experimental and computational research efforts. ICE brings computer science disciplines such as parallel processing, interactive graphics, and artificial intelligence together with concepts such as distributed computing and knowledge bases to create a unique hardware/software capability.

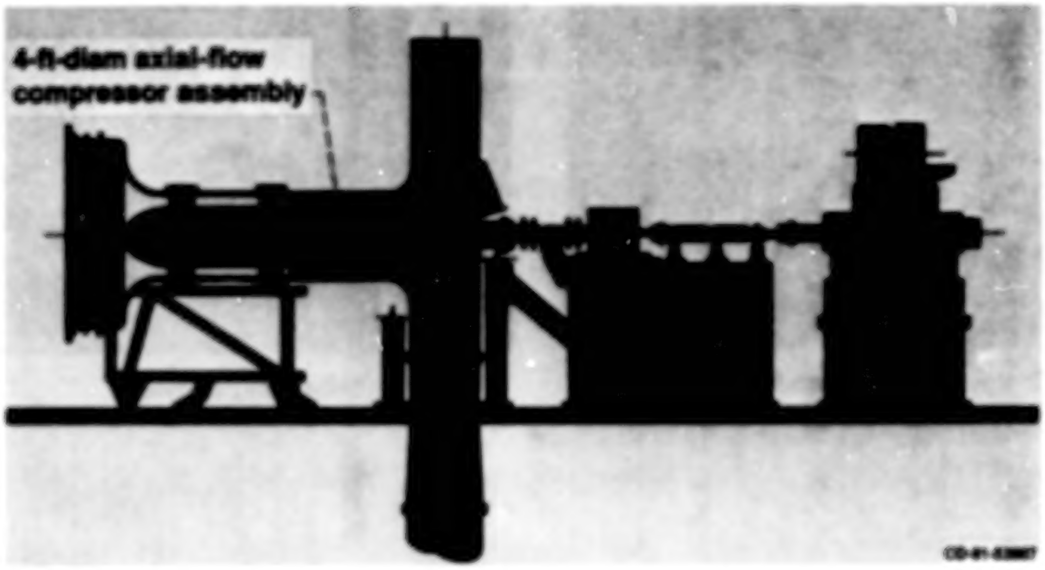
The ICE program has several key elements. The first element involves the development of methods for parallelizing CFD codes and executing them on parallel processors, which are expected to offer supercomputer performance at departmental computing costs in the near future. The second element involves using parallel processing techniques coupled with workstation graphics to rapidly reduce and display measurements from advanced instrumentation systems such as laser and hot wire anemometers. This capability will allow experimental researchers to analyze complex data sets in near real-time, while the test rig is still on-condition, and make adjustments in measurement locations or extent in order to capture flow physics details. The third element of the ICE program involves storing CFD results and experimental data as independent data bases on a common mass storage system, then retrieving both CFD and experimental results and displaying them through a common graphics interface for further off-line comparison and analysis.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Multistage Compressor Flow Physics

Low Speed Multistage Axial-Flow Compressor

Detailed mechanical design completed	1989
Fabrication and assembly	1990-1991
Operational	1992



NASA Lewis has in place a Multistage Compressor Flow Physics Program which is aimed at improving our understanding of the complex mixing phenomena which control the performance of multistage compressors. The program features a tightly coupled effort between computational, experimental, instrumentation, and high-speed-computing research areas.

The experimental research will be conducted in both high-speed and low-speed compressors. The high-speed research, focussed on the measurement of thermodynamic properties through the compressor, will be conducted in existing Lewis facilities. The low-speed research will focus on a detailed study of mixing phenomena and will be performed in a new low-speed multistage axial compressor. A four-stage axial-flow compressor research package is currently being fabricated and will be interchanged with a large centrifugal compressor impeller which is currently installed in the low speed compressor facility. The axial research package is 4 ft in diameter and rotates at 950 rpm. The large size, low speed, and long inlet result in thick end wall boundary layers, the proper blade chord Reynolds number, and minimal blockage effects associated with intrusive instrumentation. The research package is designed to enable changes in blade stagger, solidity, and relative circumferential positioning between blade rows, features both casing-mounted and rotor-mounted aerodynamic survey capability, and also features optical access for non-intrusive measurement techniques and flow visualization techniques.

Low Speed Centrifugal Compressor Research

3D Navier-Stokes code predictions of fluid particle paths

5-ft-diam impeller

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

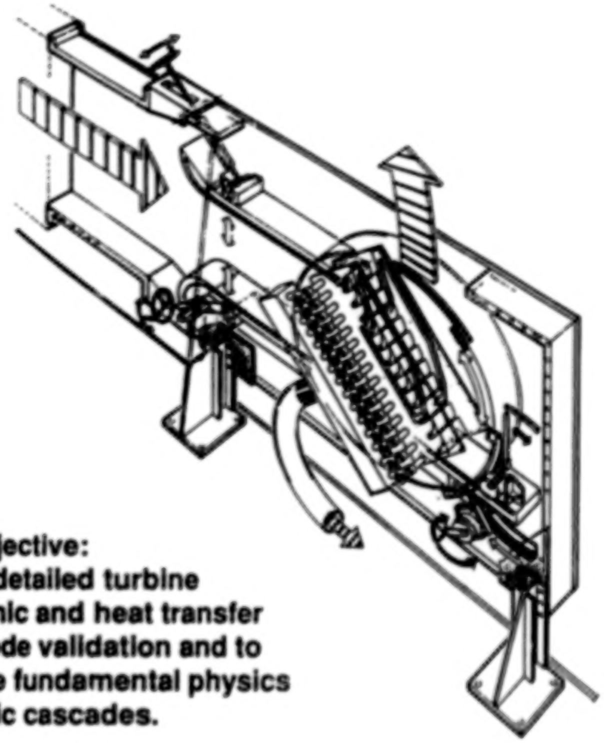
CD-91-53988

Numerical analysis of centrifugal compressor flow fields using 3D Navier-Stokes codes is being coupled with experimental measurements acquired in a unique large-size, low-speed centrifugal compressor impeller in order to improve our understanding of centrifugal compressor flow physics and to assess the accuracy of 3D viscous codes. The low-speed impeller has a tip diameter of 60 in. and a rotational speed of 1950 rpm. The large size and low speed of the impeller yield tip clearance flow features and viscous flow regions such as blade and end wall boundary layers which are large enough to measure with laser anemometry.

A photograph of the low-speed impeller is shown above, along with the results of a numerical prediction of the impeller flow field, including tip clearance effects, generated with a 3D Navier-Stokes code. The illustration above is aimed at visualizing the secondary flow field within the impeller. Fluid particles in the blade boundary layer are being tagged from hub to tip near the blade leading edge on both the blade pressure and suction surfaces and then followed as they proceed toward the impeller exit. The numerical result indicates that this fluid is being driven to the tip of the blade, then passes through the tip clearance gap, and accumulates in a vortex-like structure which exits the impeller near the pressure-surface side of blade passage. Such results improve our understanding of the complex impeller flow physics and serve as a guide in planning the location and extent of detailed laser anemometer flow field surveys.

C3

Linear Transonic Turbine Cascade



Project objective:
To obtain detailed turbine aerodynamic and heat transfer data for code validation and to explore the fundamental physics of transonic cascades.

CD-91-53989

To cut engine weight and cost, the trend in future turbine design will be toward fewer turbine stages. This means more work per stage, which means higher turning angles and transonic exit flows. Design rules based on empirical models predict that these types of blades will suffer high losses while CFD codes do not. Ever increasing turbine inlet temperatures require the accurate heat transfer predictions be made.

The test program had two objectives. First, appropriate loss models for advanced blade designs at high Mach numbers would be determined. Second, and more importantly, a database of experimental data would be obtained for code validation. Codes such as RVC3D and PROTEUS, which are designed to accurately predict the aerodynamics for transonic flow fields, would benefit from a transonic field database.

Automated Inverse Airfoil Design Code

- **Rapid generation of 2D airfoil shapes with prescribed aerodynamic characteristics.**
- **Provides baseline design configuration beyond existing data base.**

CD-91-53990

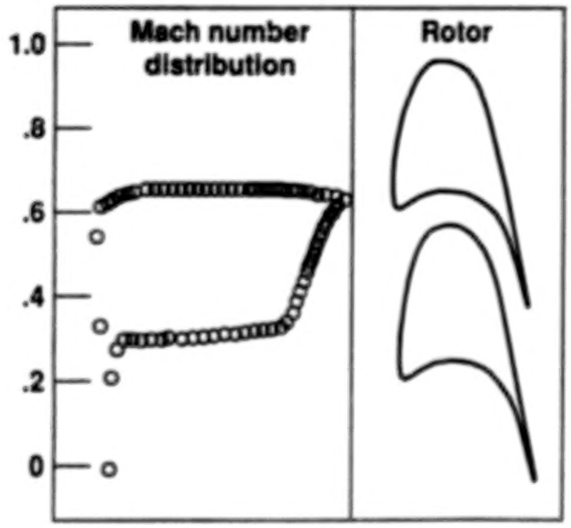
An original, innovative approach to the design of airfoils has been developed. Airfoil sections, whether for aircraft wings, turbomachinery, or numerous other applications, have historically been designed by a trial-and-error process. That is, the engineer or designer successively guesses different airfoil shapes and then, by analysis or experimentation, determines if the required performance is obtained. This performance is often measured in terms of the flow turning achieved by the airfoil and by the pressure distribution on the airfoil surface, which can be directly related to the lift and drag generated. This process is labor intensive due to the manual trial-and-error process employed.

Here at NASA Lewis, Dr. Jose Sanz has developed an automated approach that generates the airfoil shape that will satisfy the required performance as specified by flow turning and airfoil pressure distribution. This automated procedure builds on the Inverse Hodograph Method.

To further explore the range of application of the code after the incorporation of the automated procedure, which might be of interest to the engine industry in the specific field of axial turbomachinery, a set of cases has been supplied by different companies to be tried at NASA Lewis with the automated code. The information supplied consists mainly of the corresponding velocity triangles for each test case. These turbomachinery test cases include both axial compressor and turbine blading.

Automated Inverse Airfoil Design Code

- **Used to generate baseline design for an advanced turbopump turbine**
- **Design outside industry database (160° turn)**



Design parameters

Inlet Mach No.	0.548
Exit Mach No.	0.643
Inlet Flow Angle	80.11
Exit Flow Angle	–80.08
Solidity	1.372

CD-91-53991

A good demonstration of the value of the inverse design code occurred recently. The NASA Marshall Spaceflight Center had contracted with industry for the design of a generic high-pressure fuel turbopump turbine. The parameters specified for the turbine resulted in design requirements that were outside the range of industry experience. The contractor was unable to develop a baseline shape that met the requirements. Subsequently, the contractor spent two days here at Lewis working with Dr. Jose Sanz to generate the required baseline design. Since then, the contractor has been able to complete the design and release it to the computational fluid dynamics community for analysis.

CONCLUDING REMARKS

In summary, over the last few years we have seen a dramatic rise in the capability to compute the 3D flow in turbomachinery, and we have developed a program where we synergistically apply experimental and computational efforts to maximize our understanding of the complex flows existing in turbomachinery passages. This improved capability and understanding will lead to significantly better turbomachine designs in the future.

N91-20098

CHEMICAL REACTING FLOWS

Erwin A. Lezberg
NASA Lewis Research Center
Cleveland, Ohio

Edward J. Mularz
Propulsion Directorate
U.S. Army Aviation Systems Command
Lewis Research Center
Cleveland, Ohio

and

Meng-Sing Liou
NASA Lewis Research Center
Cleveland, Ohio

This paper describes the objectives and accomplishments of research in chemical reacting flows, including both experimental and computational programs. The experimental research emphasizes the acquisition of reliable reacting-flow data for code validation, the development of chemical kinetics mechanisms, and the understanding of two-phase flow dynamics. Typical results from two nonreacting spray studies are presented. The computational fluid dynamics (CFD) research emphasizes the development of efficient and accurate algorithms and codes, as well as validation of methods and modelling (turbulence and kinetics) for reacting flows. Major developments of the RPLUS code and its application to mixing concepts, the General Electric combustor, and the Government baseline engine for the National Aerospace Plane are detailed. Finally, the turbulence research in the newly established Center for Modelling of Turbulence and Transition (CMOTT) is described.

Chemical Reacting Flows

Goal:

Development of accurate predictive CFD codes which simulate the coupling of fluid mechanics and chemistry of combustors of aerospace propulsion systems

- **Experimental research to investigate the physics and chemistry**
- **Provision of a proper data set for code validation and assessment**

Impact:

- **Ability to design aeropropulsion systems with performance at least double that of current engine technology**
- **Resolution of technical issues related to propulsion of hypersonic vehicles**

CD-91-54316

The long-range goal of the Chemical Reacting Flows research program is the development and validation of predictive, time accurate, three-dimensional computational fluid dynamics (CFD) codes that simulate the coupling of fluid mechanics and combustion kinetics of combustors for aerospace propulsion systems. Drivers of the research activity are future requirements for major performance increases in hydrocarbon-fueled gas turbine engines that will operate at considerably higher pressures and temperatures than current engines, requirements for large reductions in emissions to minimize environmental impact of combustors designed for supersonic cruise, and a need to gain a better understanding of combustion chemistry and mixing interactions for hydrogen-fueled combustion systems of hypersonic vehicles.

The major activities of the program, in addition to the development of reliable time-accurate codes, are fundamental experimental and analytical research on combustion kinetics, experimental study of the interactions of fluid dynamics and chemistry, and the development of detailed reacting-flow datasets for code validation and assessment.

Hydrocarbon Combustion

- Advanced methods for reacting flow calculations
  - Numerical methods and physical models for combustion of liquid fuel sprays in a high-pressure, high-temperature environment
- Two-phase flow modeling and experiments
- Hydrocarbon kinetic mechanism development
- Chemical kinetics mechanism reduction
  - Efficient utilization of reacting-flow Navier-Stokes codes

CD-91-54017

The research in hydrocarbon combustion can be divided into three areas:

(1) Development of a fast, robust, three-dimensional Navier-Stokes code for low-speed reacting flow - including an improved physical model of the evaporation and combustion of hydrocarbon fuel sprays, advanced turbulence and mixing models, and reduced hydrocarbon kinetic mechanisms that reduce computational overhead yet adequately reproduce heat release and combustion emissions

(2) An experimental and modelling program for combusting hydrocarbon sprays to obtain detailed measurements and understanding of the physical processes of two-phase flow for development and assessment of the CFD codes

(3) Experimental measurements and modelling of hydrocarbon kinetics and mechanisms with emphasis on predicting formation of nitrogen oxides and soot (Reduced mechanisms will be developed and compared with more complete mechanisms by using flame codes and experiments.)

The research program for two-phase flow of liquid fuel sprays in air will now be shown in more detail.

Two–Phase Flow Modeling and Experiments

Objective:

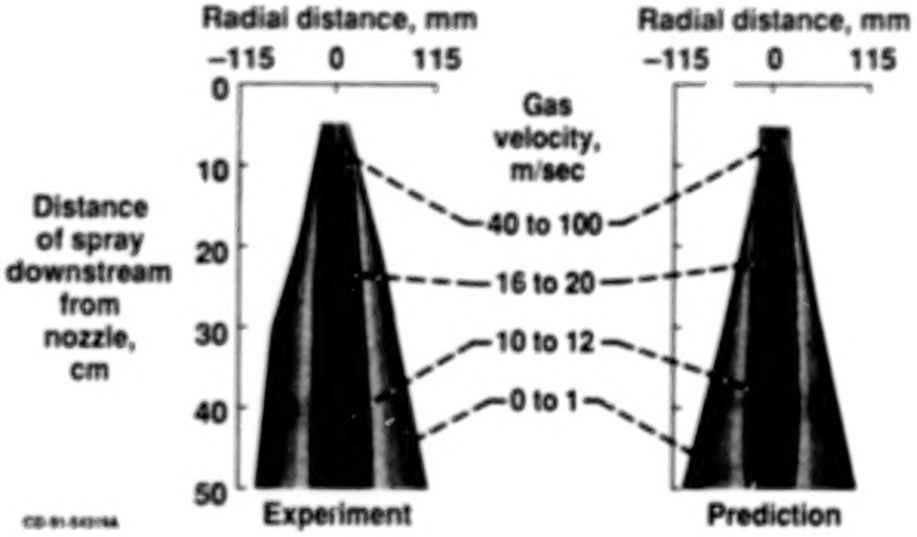
- **Perform experiments to increase understanding of physics of multiphase flows and obtain data base for validation of multiphase flow codes**

Approach:

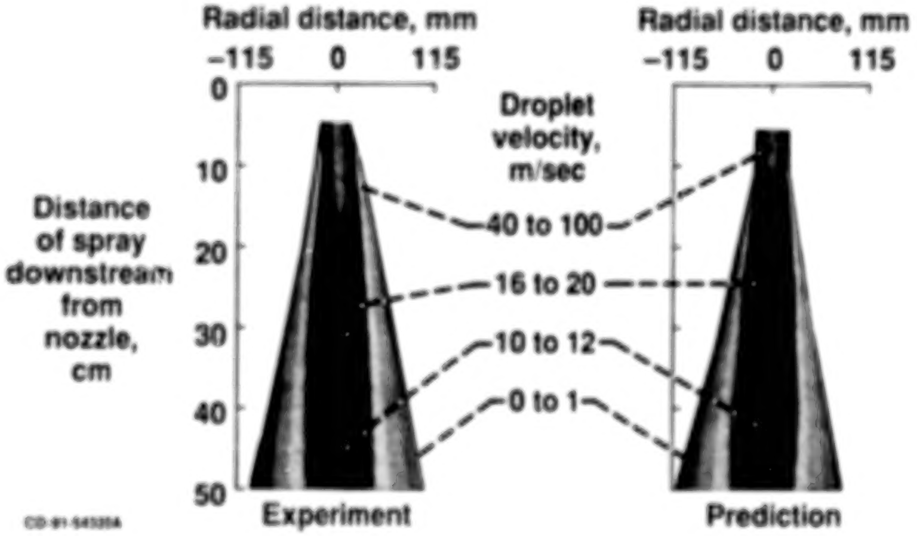
- **Perform series of experiments with increasing complexity**
  - **Particle-laden swirling flows**
  - **Nonreacting sprays**
  - **Unconfined combusting sprays**
  - **Confined combusting sprays**
  - **Pressurized combusting sprays**

CD-91-54518

The fuel-spray process is extremely important for engine efficiency, durability, and operability. The ultimate objective of this research is to develop a computer code that can accurately model fuel and air mixing with the subsequent reaction. Since this is an extremely complicated process, we are studying it in a series of steps of increasing complexity. Particle-laden jets were initially studied followed by nonreacting sprays to assess the capability of current two-phase flow models. Typical results from the nonreacting spray study are illustrated in the following figures. Currently, unconfined combusting spray experiments are being conducted and will be used to assess the capability of reacting, multiphase flow codes. Future plans are to study confined atmospheric, and then, pressurized combusting, sprays.

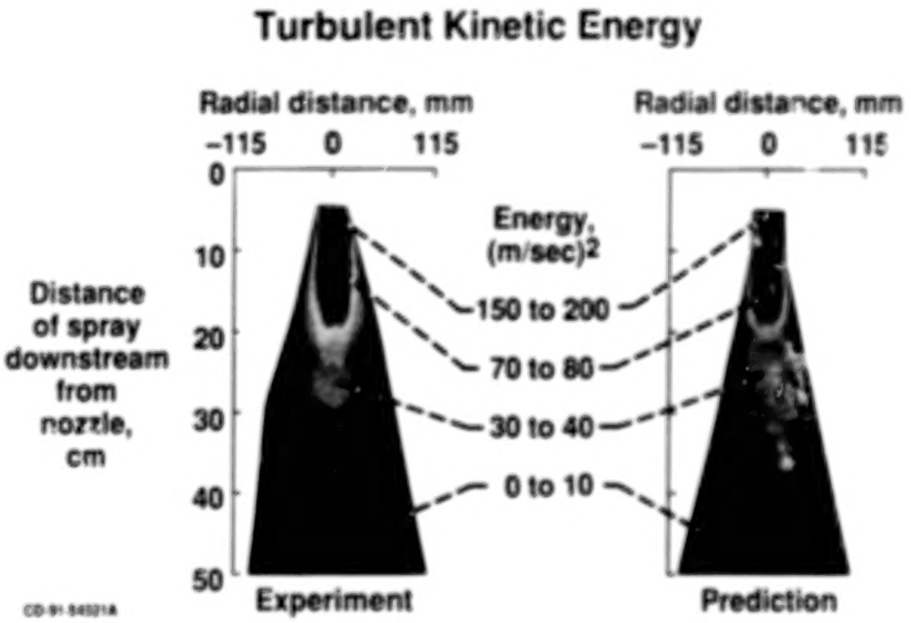


Axial Gas Velocity for Nonreacting Spray



Mean Axial Droplet Velocity (25 μm) for Nonreacting Spray

These figures compare measured and predicted axial velocities for the nonreacting spray. The experiment consisted of an air-assist atomizer spraying vertically downward into a stagnant environment. Mean axial velocities for the gas phase and 18-μm droplets are illustrated for the spray from 5- to 50-cm downstream of the nozzle. Velocity measurements were made with a two-component Phase/Doppler particle analyzer and were obtained across the entire spray. Contour levels are illustrated for velocities from 0 to 100 m/sec, and radial dimensions are in millimeters. It is evident that the spray was very axisymmetric, making it useful for model comparison. The computer model utilized for the predictions is a parabolic model with Lagrangian tracking of the particles in the computed gas-phase flowfield. Source terms are included to account for momentum exchange between the droplets and the gas phase. Turbulent dispersion of the droplets is also considered. Predictions show reasonable agreement with the data for the decay of both gas phase and droplet velocity.


Turbulent Kinetic Energy

This figure compares measured and predicted gas-phase turbulent kinetic energy in the spray flowfield. Predictions were obtained with the k-ε turbulence model. Contour levels are illustrated from 0 to 200 (m/sec)^2 for the spray flowfield from 5- to 50-cm downstream of the nozzle. Radial dimensions are in millimeters. Again, the agreement between the predictions and measurements is reasonably good.

Unconfined Combusting Spray

CD-91-54[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This photograph shows the unconfined combusting spray. The experiment consists of an air-assist atomizer spraying n-heptane liquid fuel upward into a coflowing swirling air stream. This coflow stream was swirled sufficiently by a 45° swirler to generate a recirculation zone that stabilized the flame. A thin light sheet from a copper vapor laser was used to illuminate the trajectories of the liquid fuel droplets in the combusting spray. Experimental measurements will include gas phase and droplet velocities, gas phase temperatures, and major species.

Hydrogen Combustion

Requirement:

- **Understanding of the interaction of fluid dynamics and chemistry in high-speed propulsion devices which have high subsonic and/or supersonic combustors**

Enabling technology:

- **Development of both steady and unsteady Navier-Stokes codes for analyses of combustion systems with emphasis on increased computational efficiency and improved modeling of physical processes**
- **Data sets for high-speed reacting shear layers**
- **Improved chemical model for the hydrogen-air system**

Goal:

Ability to design efficient combustion systems and accurately predict combustor performance for future hypersonic vehicles

CD-91-54323

The research in hydrogen combustion is directed toward applications to high subsonic or supersonic combustion in propulsion systems for hypersonic vehicles. Emphasis of the research program is the development and assessment of three-dimensional, reacting-flow Navier-Stokes codes which will be covered subsequently: experimental research on high-speed turbulent reacting flows to (1) improve the understanding of coupling between fluid dynamics and combustion, and (2) establish a dataset to validate computer codes which simulate the physics and chemistry of high-speed chemical reacting flows. Experimental and analytical studies of hydrogen-air kinetic rates and modelling are also being conducted which are relevant to ignition, kinetics losses, and ground test facility simulation issues for supersonic combustion.

Hydrogen Combustion– Program Elements

- **Planar reacting shear layer**
  - **Experimental database for CFD code validation**
  - **Understanding of turbulence-chemistry interactions**

- **Hydrogen-air kinetics rates and mechanism development**

- **Computational studies of the Ram-Accelerator**
  - **Propulsion and test facility applications**

CD-91-54324

Research in hydrogen combustion is directed in three areas:

(1) A planar reacting shear layer experiment is being studied to obtain steady-state reacting hydrogen data of mean and fluctuating velocities and concentrations, to compare benchmark data with predictions from computer models, and to evaluate turbulence-chemistry interactions. The experiment is designed to mix and react two streams downstream of a splitter plate. One of the streams is hydrogen that is diluted with nitrogen and the other is air that is preheated up to 870 K. Flow rates are continuous at high subsonic velocities. The facility is now operational, and Mie scattering data are being obtained for initially nonreacting streams.

(2) Reaction rates for hydrogen-air are being obtained in a chemical shock tube that uses laser absorption diagnostics to measure hydroxyl radical concentrations. The objective is the determination of three-body recombination rates of hydrogen and oxygen and the effect of water on these rates. Radical and atom recombination rates are particularly important in determining kinetic losses in exhaust nozzles of hypersonic vehicles.

(3) A computational program is in progress on the Ram-Accelerator concept to supplement experimental work at the University of Washington. In this concept, the centerbody of a shaped projectile is fired into a tube containing premixed fuel and oxidizer. Combustion occurs in either a subsonic, supersonic, or stabilized detonation mode between the projectile and the tube, creating a high-pressure region behind the projectile to accelerate the projectile to hypersonic velocities. The computational analysis is carried out with the fully coupled, Reynolds-averaged Navier-Stokes equations. Results include shock-boundary interactions and forebody precombustion effects that give credence to the process.

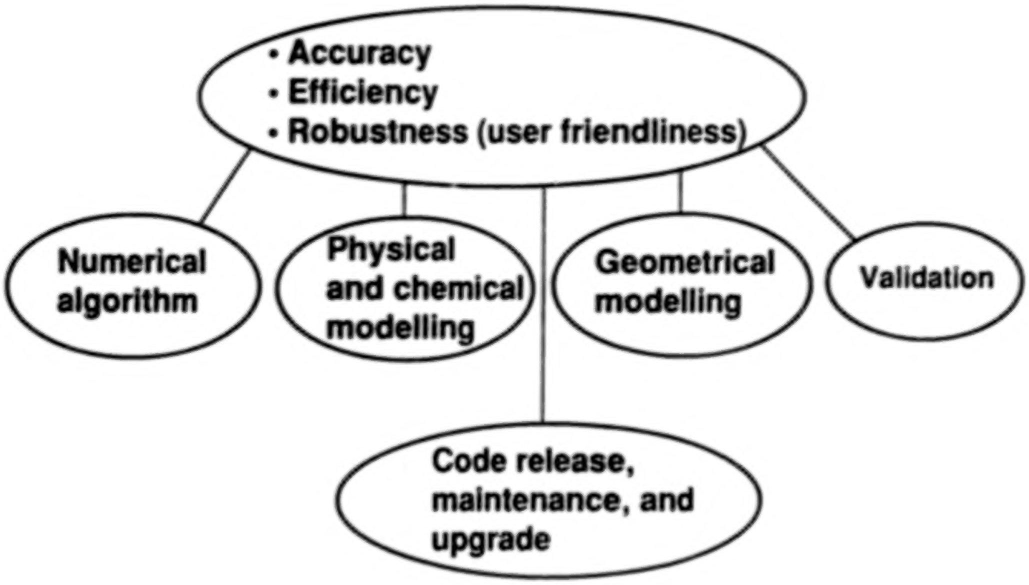
Chemical Reacting Flows— CFD Goals

- Develop reliable, versatile, 3-D CFD codes for chemical reacting flows for practical geometries and conditions
- Support national programs and industry needs

CD-91-54325

Our CFD goals for chemical reacting flows are shown above. Here at NASA Lewis Research Center, we are responsible for developing three-dimensional CFD codes that are reliable in accuracy and are versatile - able to accept complex geometries and flow conditions. The codes are designed to efficiently solve equations describing viscous, finite-rate chemical reacting flows. Another objective is to closely support national programs and industry needs in aeropropulsion systems, such as the National Aerospace Plane (NASP) and the High-Speed Civil Transport (HSCT). In order to accomplish these goals, a substantial emphasis is placed on the following issues related to CFD.

Issues of CFD



- Accuracy
- Efficiency
- Robustness (user friendliness)

Numerical algorithm

Physical and chemical modelling

Geometrical modelling

Validation

Code release, maintenance, and upgrade

CD-91-54326

This chart describes the essential requirements that a CFD code must take into consideration: namely accuracy, efficiency, and robustness (or user friendliness). Whether or not these requirements can be successfully met critically depends on the following building blocks: (1) the numerical algorithm, (2) the physical and chemical models, (3) the geometrical model, and (4) validation. All of these have been under intensive research and development in the past 20 years and are still evolving to meet the three requirements.

Here at Lewis, we are studying every aspect of these building blocks. For the numerical algorithm, we have developed a unique, efficient algorithm which is especially suited for solving a large system of equations in chemical reacting flows. The resulting code is called RPLUS and its various versions have been released to many institutions in the country. This paper describes three studies carried out at Lewis and General Electric (the latter, a case of close collaboration between Lewis and industry). We are also looking at some forefront numerical methods to improve accuracy and efficiency. In physical and chemical modelling, and especially in turbulence modelling, a strong and focused group, the Center for Modelling Turbulence and Transition (CMOTT), has been at the Institute for Computational Mechanics in Propulsion (ICOMP) at Lewis for over a year. This group has been taking rigorous approaches, ranging from accessing effects of numerical dissipations to developing new models for treating effects like compressibility and turbulence/chemical interactions. In geometrical modelling, we are looking at surface- and field-grids generation, as well as multiblocking and adaptation in the CFD codes. Finally, the ultimate test is the validation of the code. This never-ending process requires a more in-depth comparison as measurements provide more information and as physical models become more sophisticated. The research and development in CFD are ever-evolving, and these requirements will only become more stringent. Our CFD goal, ultimately, is to release and maintain codes developed at Lewis. The codes will be continually upgraded as building blocks are improved. In the rest of this paper, I will discuss the progress in these areas.

Development of RPLUS Code

- Development under the NASP TMP (1988 to 1990), and support continued under generic hypersonic program
- Code released to U.S. industry, universities, and government labs (over 40 releases)
- The code currently has the following capabilities:
  - 3-D, Navier-Stokes equations with finite-rate chemical reactions
  - Multiblock decomposition for geometrical flexibility
  - Baldwin-Lomax and k-ε models
  - Central and upwind differencing

CD-91-54327

The development of the RPLUS code was originally funded under the NASP Technical Maturation Plan (TMP) which began in 1988 and ended in 1990. Starting in 1991, the funding was picked up under NASA's generic hypersonic program. Various versions of the code have been released to requesters from U.S. industry, academia, and government labs, totaling over 40 copies. One workshop was held in the fall of 1988. We have designated contact persons for handling the transmission as well as for answering questions regarding the code. Several requesters have spent time at Lewis becoming familiar with the intricate details of the code and the theory. We have also incorporated some valuable input from the users. Such two-way dialogues were found to have great mutual benefits.

The code solves three-dimensional, Navier-Stokes equations coupled with finite-rate chemical kinetics equations. In the case of H_2-air reactions, 9 species and 18 reaction steps were considered, resulting in 13 conservation equations, rather than 5, to be solved. The chemical kinetics models are user-supplied and not hard-wired as an integral part of the code, thus giving flexibility. For complex geometries, it is difficult to generate a single grid without introducing distortion, which usually, in turn, reduces accuracy and stability. To avoid distortion, we adopt a multiblock approach - decomposing the entire domain into several blocks for which it is easy to generate a good quality grid. The global conservation is enforced in the treatment of interfaces. In addition to the Baldwin-Lomax model, the code includes several variations of the κ-ϵ model. An upwind version of the code is also available.

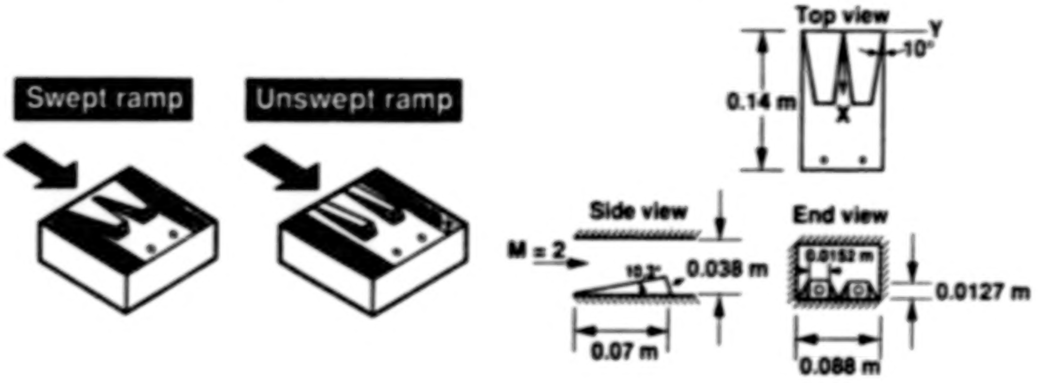
Representative Applications
of
RPLUS Codes

- Swept and unswept fuel injectors
- Government baseline engine
- General Electric combustor

CD-91-54328

Next, we present some representative applications of the RPLUS code to demonstrate its capability and to ultimately shed some light on the fluid dynamic processes of design concepts. The calculations for the first two cases were carried out at Lewis by Young-June Moon and Beverly Duncan, respectively. The third case was performed by Mani Subramanian of GE and his group, in close collaboration with Peter Tsai and Tawit Chitsonboom. This case especially illustrates how much one can achieve by using a nondesign code to give design parameters.

Swept and Unswept Fuel Injectors



Incoming Stream

Mach number	2
Pressure, Pa	102 000
Temperature, K	1024
Mass fraction	
H_2O	0.182
O_2	0.256
N_2	0.562

Hydrogen Jet

Mach number	1.7
Pressure, Pa	325 200
Temperature, K	187
Equivalence ratio	1.2

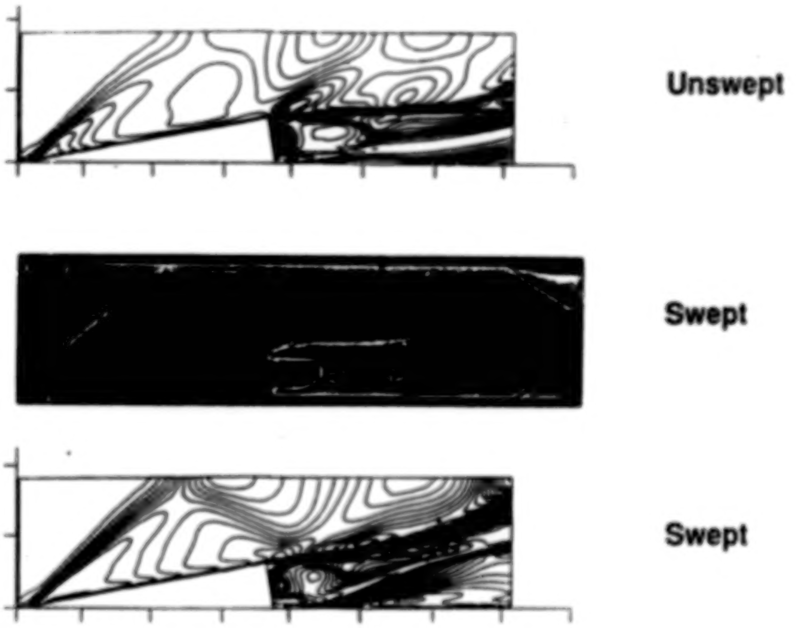
CD-91-54329

This figure shows the configuration of the fuel injectors for a supersonic combustor investigated experimentally by Northam et al. of NASA Langley Research Center. Since mixing is a crucial step for initiating chemical reactions and the mixing is diminished in high-speed streams, this configuration is used to study concepts to enhance mixing via generating streamwise vorticity. The flow conditions are given above.

Because of the ramp and corner topologies and the requirement of the fine grid at the surfaces, a single grid gives rise to undesirable distortion and nearly singularities at the corners. These deformations generally lead to nonconvergence and excessive numerical dissipation. A multiblock strategy was used to overcome this difficulty, and as a result, the calculation became easy to handle. The grid densities are 0.16 and 0.17 million, respectively, for unswept and swept cases, each using three blocks.

Swept and Unswept Fuel Injectors (continued)

Density Contours

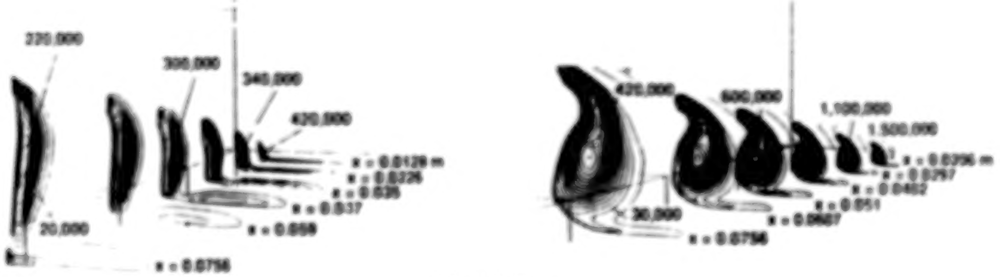


CD-91-54330

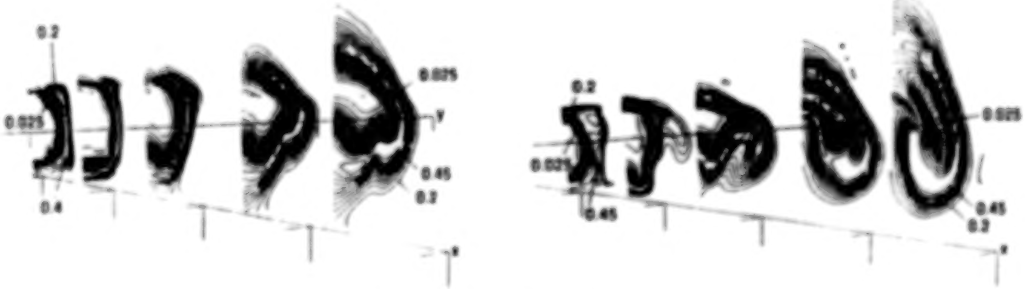
Here we show the density contours of both swept and unswept configurations at the jet centerplane and the comparison with the shadowgraph for the swept case. Qualitative features such as the shocks, expansion wave, fuel jet plume, and shear layer are well represented, despite the relatively coarse meshes utilized. Simply because of the difference in geometry, two effects are noticeable. In the swept case, because the leading edge is two dimensional, the ramp shock is stronger, and the subsequent reflected shock from the top wall remains stronger. Second, the fuel jet is lifted away from the ground more because of the induced motion by the stronger vorticity generated along the ramp edge.

Swept and Unswept Fuel Injectors (concluded)

Vorticity and H_2O Contours



Vorticity

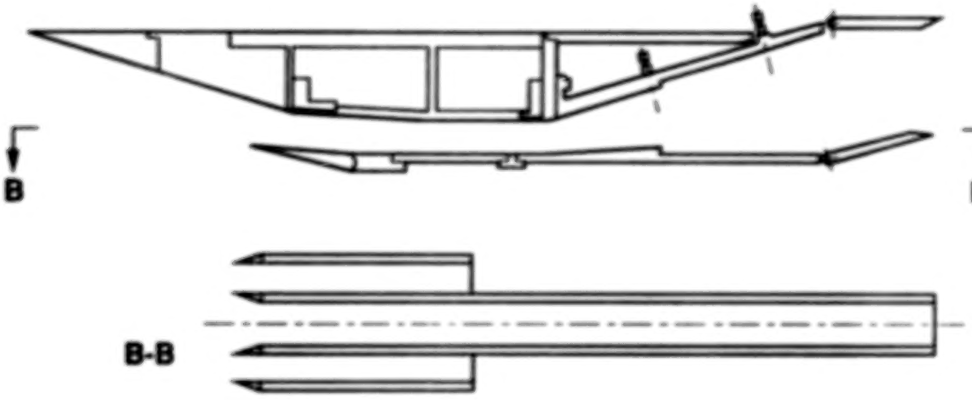


H_2O Mass Fractions

CD-91-54331

The vortical structure generated by the spanwise nonuniformity is clearly displayed at five cross planes upstream of the injector face. In the swept case, this vortical structure has much greater strength and grows in three dimensions, while in the unswept case, the spanwise growth is rather confined close to the ramp side wall. The water vapor concentrations are compared at five planes downstream of the injector. The fuel jet is seen more rapidly deformed and its core is merged quickly into the core of the vortex generated upstream at the ramp. Finally the core moves away spanwise from the jet centerplane, while the core of the unswept case is deformed but still remains at the centerplane. Thus, it is important that this vortical structure be generated and that its major growth take place sufficiently ahead of the injection so that the mixing of the two streams can be effective. This clearly shows the advantages of using the swept injector for promoting mixing and, thus, combustion.

Government Baseline Engine



Free-Stream Air

Mach number	3.44
Total temperature, K	750
Total pressure, atm	4.2

Hydrogen Jet

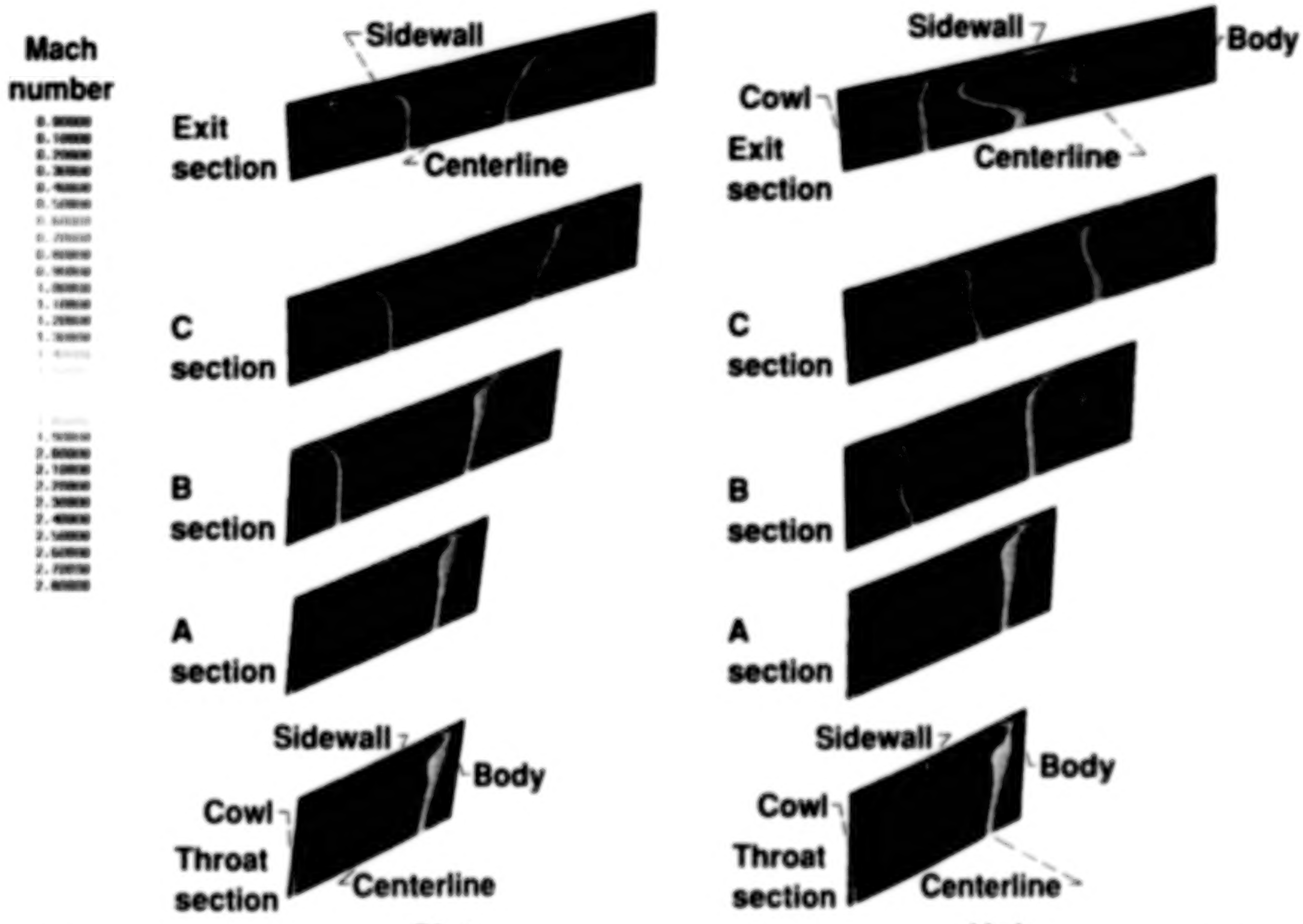
Mach number	1.0
Temperature, K	293
Equivalence ratio	0.4

CD-91-54232

This sketch shows the Government baseline engine model used to study hypersonic propulsion concepts. The experiment was planned and carried out by Dave Sagerser of Lewis. Then, the test was performed initially at General Applied Science Laboratories (GASL) and subsequently at Lewis. There is a synergistic coordination between the experimental and computational groups. An objective of the computational effort is to demonstrate the current CFD capability for calculating a complete "nose-to-tail" configuration, using a complicated set of chemical kinetics. We also studied the effects of slot and discrete fuel injectors on mixing and combustion, and valuable information can be extracted for examining new concepts. This case shows how CFD has become an indispensable and cost-effective tool in various phases of a design cycle, despite uncertainties regarding the previously mentioned building blocks, such as modelling and validation. The flow conditions are listed above, and laminar flow is assumed. The total mass of injection was kept the same to compare the effectiveness of the two injections; thus different injection pressures were required.

Government Baseline Engine (continued)

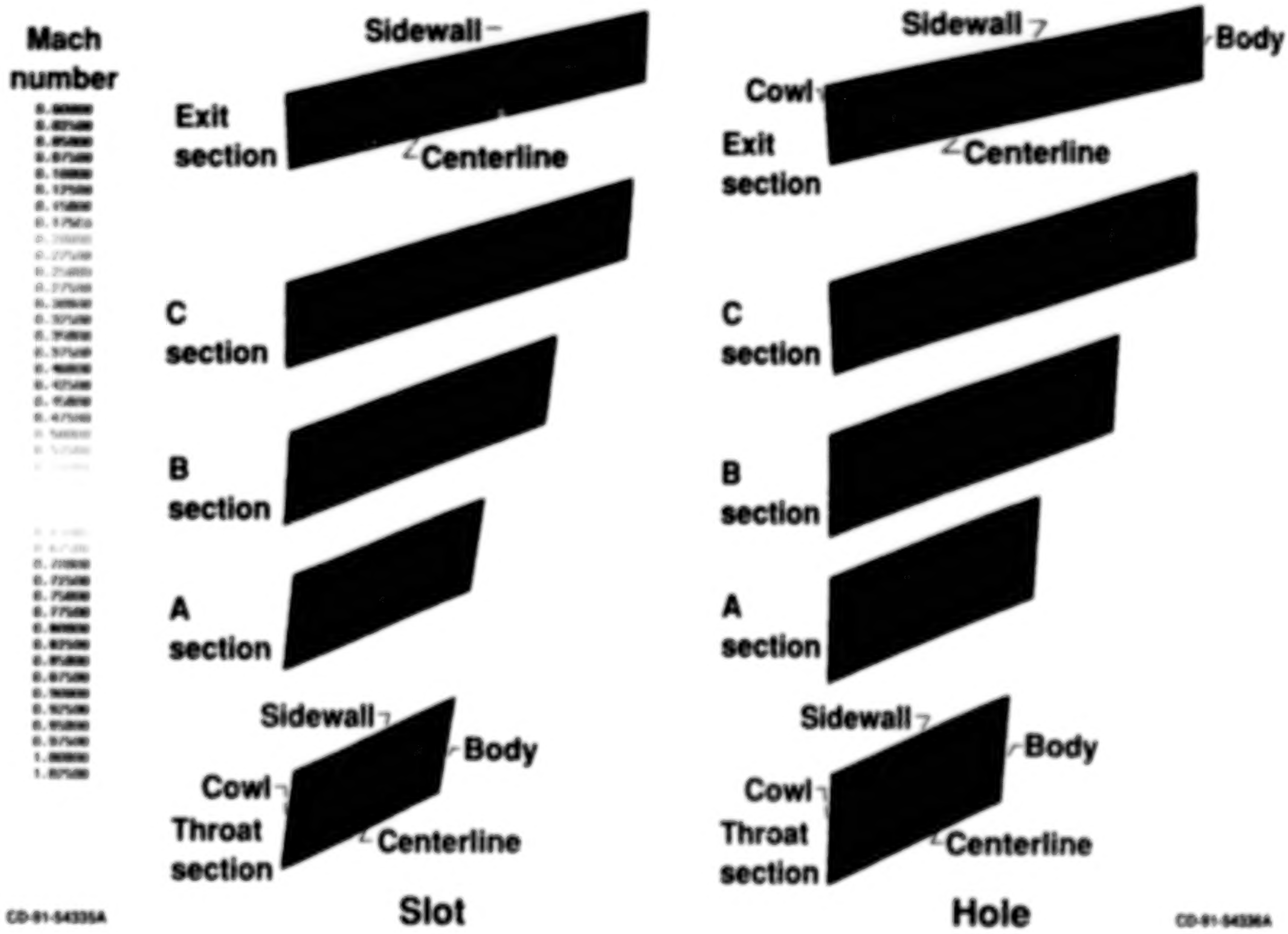
Mach Number Contours



This chart displays the Mach contours with the slot and discrete hole fuel injections in the engine. Five transverse planes, four located downstream of the injection port and one at the throat, were plotted to indicate the extent of mixing due to these two injections. The inviscid core and the low-momentum region near the walls are clearly visible. It is clear that the discrete injector provides much greater mixing since it generates a spanwise gradient. Thus streamwise vorticity is generated, and it, in turn, induces cross-plane velocity components which are responsible for enhancing the mixing. It is noted that the body side has better mixing than the cowl side. This is attributable to the thicker boundary layer initiated by the cowl shock and convected downstream. In the case of hole injection, a kidney-shaped secondary flow containing high vorticity is evident, and the nonuniformity at subsequent planes becomes increasingly stronger, leaving a greatly reduced inviscid core at the exit.

Government Baseline Engine (concluded)

H_2O Contours



CD-91-54335A

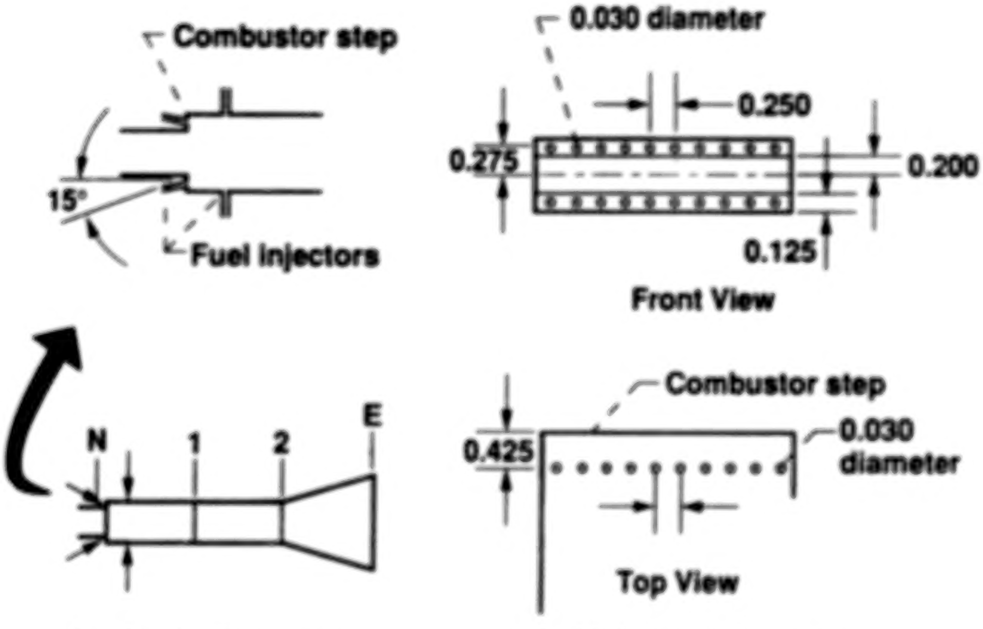
CD-91-54336A

Here we show the extent of reaction due to the two injections. As expected, the higher the mixing, the better the combustion. A dramatic difference in the H_2O concentration clearly indicates the effectiveness of the hole injection, while the slot injection only allows the reaction to take place in limited regions. In both cases, since the air entering the combustor (even at adiabatic temperature) is at too low a temperature to ignite, a "torch ignitor" is placed in the computation.

The computational results will give extremely useful guidelines to researchers planning experiments, particularly in such complicated fluid dynamic situations where little information is known a priori. The experimental data are being generated and processed. Detailed comparison will follow and should clarify the areas that need further investigation. Currently both efforts of test and computation are continuing.

General Electric Combustor

- Dimensions are in inches


Combustor step
15°
Fuel injectors
0.030 diameter
0.250
0.275
0.200
0.125
Front View
N
1
2
E
Combustor step
0.425
0.030
diameter
Top View

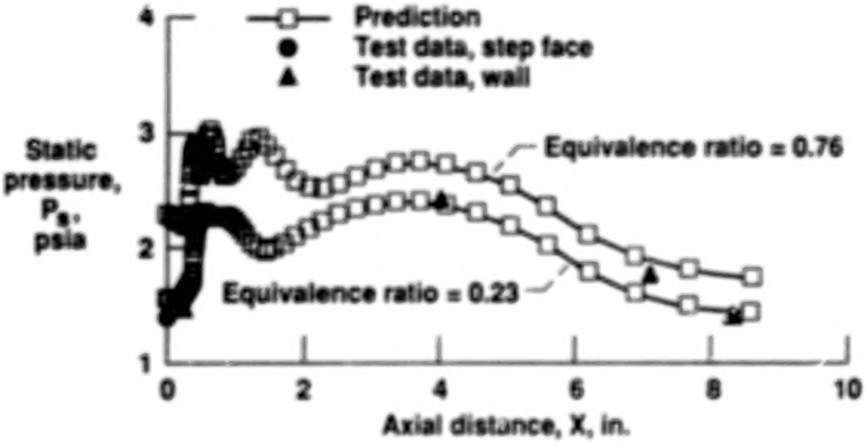
Combustor Geometry

Fuel Injector Schematic

CD-90-54327

This figure shows the geometry of the GE scramjet combustor which was measured at the Ohio State University. A very extensive interrogation of the RPLUS code was performed by the GE group led by Subramanian. A detailed study of flow physics and mixing in this combustor was conducted for Mach numbers from 3 to 6. A number of injection geometries and arrangements were investigated. The flow conditions are shown in the figure. The Baldwin-Lomax model was extended to account for three-dimensional corner effects. The heat releases of various configurations were compared, and an extensive set of design parameters were examined to assess the performance. Also shown are the geometries of the injectors, including the circular hole and the normal and slanted slit-slot.

General Electric Combustor (continued)
Particle Traces and Wall Pressure



CD-91-54238

Scramjet Combustor Geometry
Particle Traces

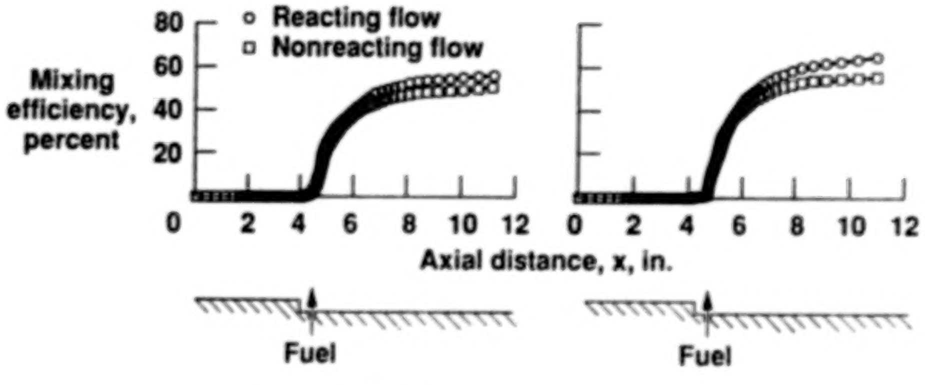
[Equivalence ratio = 0.76]

CD-91-54239

Shown in this chart are the three-dimensional view of particle traces and the pressure distribution along the bottom wall at midway between normal jets. The test was performed at the Ohio State University with an upstream Mach 3 condition. Few data points that are available appear in good agreement with the calculations. The particle traces also clearly show the vortical structure and the entrainment of free stream fluid into a spiral motion. However, at about twice the step-height the stream does not mix with the injected hydrogen anywhere in the combustor. Next, various injection concepts are examined to assess their performance.

General Electric Combustor (concluded)
Injector Geometries and Mixing Efficiency

[Equivalence ratio = 1.5, Mach number = 18, dynamic pressure ratio = 2.0]



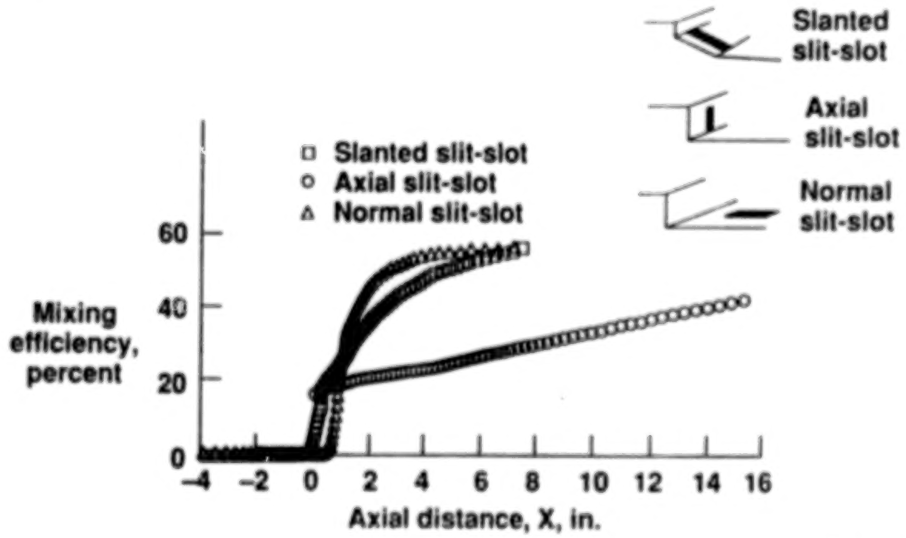
Circular Hole Injector

Slit-Slot Injector

CD-91-54340

Mixing Efficiency Versus Axial Combustor Length

[Equivalence ratio = 1.5, Mach number = 18, dynamic pressure ratio = 2.0]



CD-91-54341

On this page we compare mixing efficiencies of several injector configurations as well as the effect of the chemical reaction. The incoming conditions corresponding to a Mach 18 flight are M = 6.7, P = 7.05 psia, and T = 3246 °R. The fuel injector has conditions at M = 2, P = 126 psia, and T = 1123 °R. First the slit-slot injector appears to be about 5 to 10 percent more efficient than the circular injector, and the H_2-air chemical reaction actually promotes mixing. Thus, we focus now on varying the locations of the injector - at the bottom wall, the step face, or the slanted face. The bottom wall (normal) injection is the most efficient, and the axial injection at the step face is the least efficient. The collaborative effort between GE and Lewis is continuing, and the CFD information will continue to assist design modifications and performance improvements studies.

Turbulence Modelling

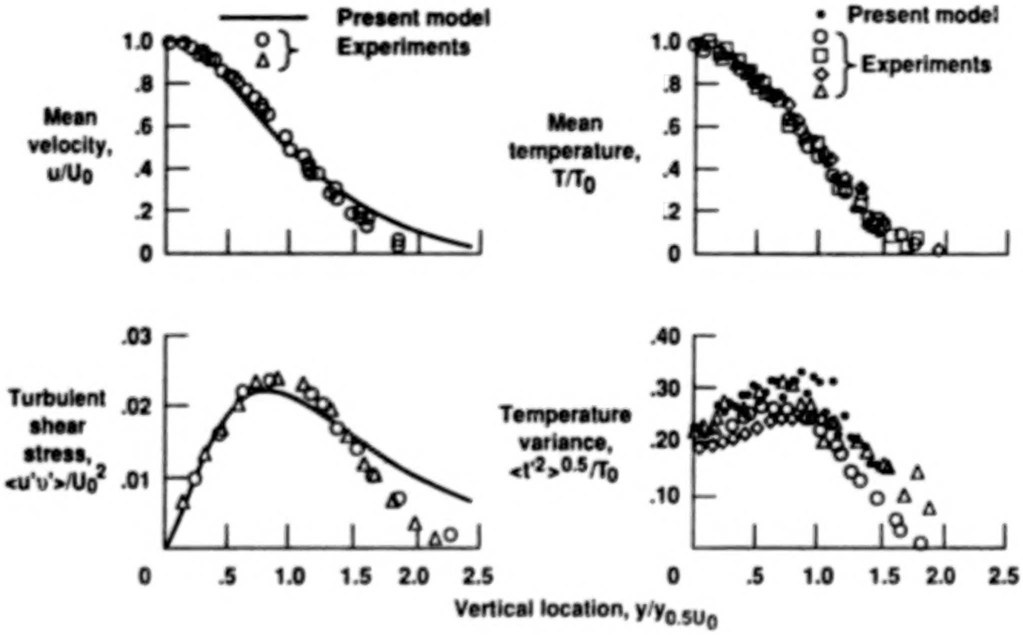
- **PDF turbulence combustion modelling**
- **Development of k-ε and second-moment models**

CD-91-54342

As I mentioned earlier, there has been considerable progress in turbulence modelling made by the CMOTT members at Lewis. We have been looking at using the probability density function (pdf) approach to represent the turbulence fluctuations due to chemical reactions. In the past year, significant effort has been spent on assessing the currently available two-equation models and second-order models. In the next two pages, I present only two of these cases to show some of the significant results.

PDF Turbulence Modeling

Heated Turbulent Free Jet

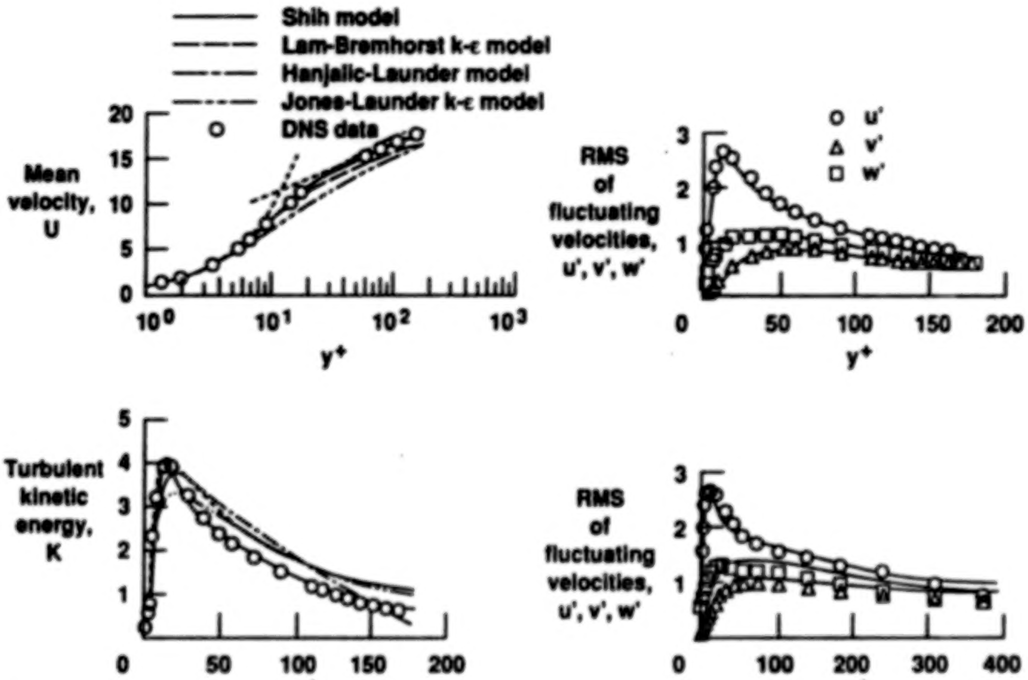


CD-91-54343

This figure shows calculated solutions of a heated turbulent free jet using a pdf model by Andrew Hsu. Good agreement of mean velocity and temperature distributions with experimental data is obtained. The correlations of fluctuating quantities are also included for comparison. Except a slight over-prediction near the edge of the jet, the numerical Reynolds shear stress generally agrees well with the data. An accurate prediction of the Reynolds stress terms is essential for the correct modelling of molecular mixing in the pdf calculation. Also, the calculated temperature standard variance agrees very well with the experimental observations. An accurate representation of temperature fluctuations is particularly important in chemical reactions because the reaction rate is strongly dependent on the instantaneous temperature.

Second-Moment Turbulence Modelling

Channel Flow



CD-91-54344

This page shows the results of using higher-order turbulence models for channel flow. T.-H. Shih, who serves as the technical leader of CMOTT, has been developing second-moment closure and κ-ϵ models. Although theoretical justification has been employed to the maximum extent possible in the development of turbulence models, experimental observation and measurement have also been indispensable. With the rapid growth of computer power, there is a new source of information: the direct numerical simulation of the Navier-Stokes equation is now possible, but it is limited to low Reynolds numbers. This figure compares results from two κ-ϵ models and two second-moment models with the direct numerical simulation (DNS) data. Shih's model, which is indicated by the solid line, has produced both mean and fluctuating results in excellent agreement with the DNS data, in particular the peaks show excellent agreement. Further efforts in compressible turbulence model development and verification for more complex flows for the second-moment closures are well underway.

Current and Future Research

- Development of advanced numerical algorithms
- Development of compressible k-ε and second-moment models
- Incorporation of interactive (menu-driven) grid generation in the RPLUS code
- Continuing validation of the code and models
- Continuing support of onsite and offsite research and development

CD-91-54345

This chart summarizes some of the current and future research in terms of the four building blocks described earlier: namely, in the areas of developing advanced numerical algorithms, physical and geometrical models, and validations. Research on new algorithms has resulted in several promising possibilities. Also, significant collaboration with ICOMP researchers has been maintained. Now, compressible models are being developed by taking the direct interaction (DIA), renormalization group (RNG), and second-moment approaches. Research on using menu-driven ideas for interactively generating grids has been conducted in the last few years, and Yung Choo and Peter Eiseman plan to implement this research into the RPLUS code. As stated over and over again, the validation of the RPLUS code and models is continuing by examining more complex flows and more complete and reliable data sets. Last, but not least, we fully believe in supporting relevant onsite and offsite programs and welcome the opportunity of participation.

N91-20099

INSTRUMENTATION AND CONTROLS OVERVIEW

Norman C. Wenger
NASA Lewis Research Center
Cleveland, Ohio

SUMMARY

The Lewis Research Center has had a long history of research directed toward advancing the Nation's capability in the areas of propulsion research instrumentation and propulsion controls. This session will highlight some of the major advances from this research as well as highlight some of the planned research that will strongly impact our future capabilities. The presentations will cover our efforts on research instrumentation and controls as well as our efforts in high-temperature electronics.

This introductory section will focus on the goals, scope and major thrusts in each of the research areas.

Instumentation and Controls Research

- Propulsion instrumentation
- High temperature electronics
- Controls

CD-91-53878

The Lewis Research Center's Instrumentation and Controls Research Program consists of three main elements: propulsion instrumentation research, high-temperature electronics research, and controls research.

Propulsion Instrumentation Research

Goal: Provide the instrumentation technology advances needed to support future aeropropulsion research and development

Scope of applications:

Fundamental studies

Component R&D

Propulsion system R&D

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91 53679

The goal of the Propulsion Instrumentation Research Program is to provide the instrumentation technology advances needed to support future aeropropulsion research and development. Instrumentation is being developed for a wide range of applications which include fundamental studies of basic phenomena, propulsion system component research and development, and propulsion system research and development. Examples shown include fundamental studies of combustion phenomena in a flat-flame burner, detailed measurements of flow in a turbine stator cascade, and propulsion system tests using an advanced engine in a altitude simulation chamber.

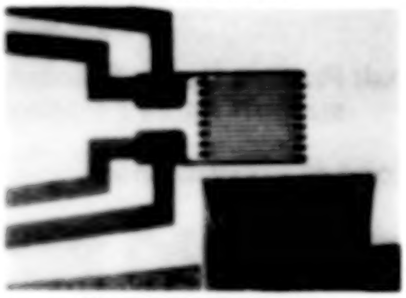
Propulsion Instrumentation Research

Thrusts:

Develop measurements capabilities

- **For use in hostile propulsion system environments**
- **For use in conjunction with advanced materials (e.g., ceramics, composites)**
- **That provide information over 2-D or 3-D regions**
- **That provide multiparameter measurements**

Research focus:



Contact sensors

Remote optical measurement systems

CD-91-[illegible]

The Propulsion Instrumentation Research Program includes many thrusts. Some of the major thrusts include the development of measurement capabilities for use in hostile propulsion system environments and for use in conjunction with advanced propulsion system materials such as ceramics and composites. Other thrusts focus on the development of measurement systems that provide information over a two-dimensional or three-dimensional space and on measurement systems that provide multiparameter measurements. The "bottom line" thrust is to develop the measurement capabilities that will provide the quality data needed to accurately assess performance and to validate a wide variety of aero-thermal-structural codes.

The research focus includes two main types of instrumentation: Contact sensors and remote optical measurement systems. The Lewis program gives equal emphasis to the two types of instrumentation. Examples shown include a thin-film strain gage on a ceramic substrate and an optical system for making remote measurements of strain on a test specimen. Both instruments are designed for making static strain measurements at high-temperature.

High Temperature Electronics Research

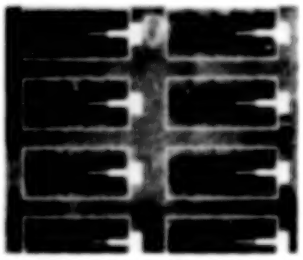
Goal: Provide the solid state semiconductor technology needed for a wide variety of high temperature integrated sensors and other electronic devices

Research focus: Silicon carbide based technology

Program status:

Today

MOSFET Array



Discrete devices

Mid 1990's

Integrated sensors

Late 1990's

Electronic systems

CD-91-53881

The goal of the High Temperature Electronics Research Program is to provide the solid state semiconductor technology needed for a wide variety of high-temperature integrated sensors and other electronic devices. The research focus is on silicon carbide since it has a potential maximum operating temperature capability far higher than the 300 °C capability of silicon.

The program has made significant progress in the past few years. Today, we have demonstrated in the laboratory discrete semiconductor devices such as a Metal-Oxide-Semiconductor-Field-Effect-Transistor (MOSFET) operating at temperatures of 500 °C. Our goal by the mid 1990's is to demonstrate integrated electronic sensors, and by the late 1990's we hope to demonstrate high-temperature electronic systems.

High Temperature Electronics Research

Thrusts: **Silicon carbide semiconductor crystal growth and characterization**

Electronic device fabrication (e.g., diodes, transistors)

Crystal growth

Characterization

Device fabrication

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

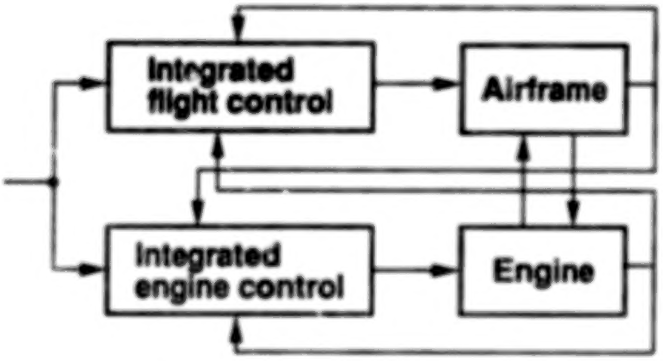
CD-91-53882

The High Temperature Electronics Research Program currently has two major thrusts: Silicon carbide semiconductor crystal growth and characterization and electronic device fabrication. Strong emphasis is placed on the crystal growth and characterization efforts since the production of electronic devices is critically dependent on the availability of large-size and defect-free crystals. Shown are a chemical vapor deposition system for growing the crystals, an energy dispersive x-ray analyzer system for characterizing the crystals, and an optical mask aligner employed in the device fabrication process.

Controls Research

Goal: Provide the advanced control technologies (i.e., control hardware and design methodologies) needed to support a wide variety of future aircraft and propulsion systems

Research scope:



Advanced control design

Control sensor R&D

Flight validation

CD-91-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The goal of the Controls Research Program is to provide the advanced control technologies needed to support a wide variety of future aircraft and propulsion systems. Primary emphasis is on propulsion controls. The program scope includes control sensor research and development, design of advanced control systems, and flight validation of advanced controls. Shown are a block diagram of an integrated propulsion/airframe control system, an optical angular position encoder for use in a fly-by-light control system, and the F-18 aircraft on which we plan to conduct flight testing of a fly-by-light propulsion/airframe control system.

Controls Research

Thrusts:

- **Fly-by-light controls technology**
- **NASP propulsion system dynamics and controls**
- **ASTOVL integrated controls**
- **Life extending controls**

CD-91-53884

Our Controls Research Program contains numerous thrusts. Listed are those which we will highlight in this conference. As the list indicates, our efforts cover a very broad spectrum.

N91-20100

PROPULSION INSTRUMENTATION RESEARCH

William C. Nieberding, Daniel J. Lesco, and W. Dan Williams
NASA Lewis Research Center
Cleveland, Ohio 44135

As was stated in the overview, "the goal of the Propulsion Instrumentation Research Program is to provide the instrumentation technology advances needed to support future aeroproplsion research and development."

A hallmark of this work is that the sensors and measurement systems being developed are not intended to be used on operational propulsion systems. They are aimed at experiments for engine development, component development, and analytical code validation. Although sensors and/or systems for operational engines sometimes grow out of this work, they are not the goal.

A further characteristic of this work is that it is frequently blind about whether the application is for aero or space propulsion. Some of the following examples are currently being developed for space propulsion systems, but they are also applicable to aeropropulsion.

Propulsion Instrumentation Research

Goal: To develop the sensors and measurement systems required to determine component performance and to validate analytical codes

Drivers:
- Hostile environments
- New materials (primarily ceramics)
- Need for spatially and temporally resolved data for codes

Approach:
- Contact sensors (minimally intrusive)
- Optical measurement systems (nonintrusive)

CD-91-54216

Propulsion Instrumentation Research is driven by the need to determine component performance and to validate analytical codes.

The goal of much current aeropropulsion work is to increase performance. This is leading to higher temperatures and generally more hostile environments, which in turn, is leading to the use of new materials such as ceramics.

The cost of cut and try development has become prohibitive, so design data obtained via codes and via component testing must be as detailed and as credible as possible. Code validation requires that data be obtained with very high spatial and temporal resolution; in addition, it must be obtained in a nonintrusive manner since its accuracy and applicability could be compromised by the presence of a sensor that perturbs the very measurement being made.

Our response to these problems has been to work on contact sensors and optical measurement systems. The contact sensors are usually thin sputter-deposited films for thermocouple, strain gages, and heat flux gages. The optical systems usually feature a laser beam, or beams, and complex methods of data collection and processing.

Propulsion Instrumentation Research

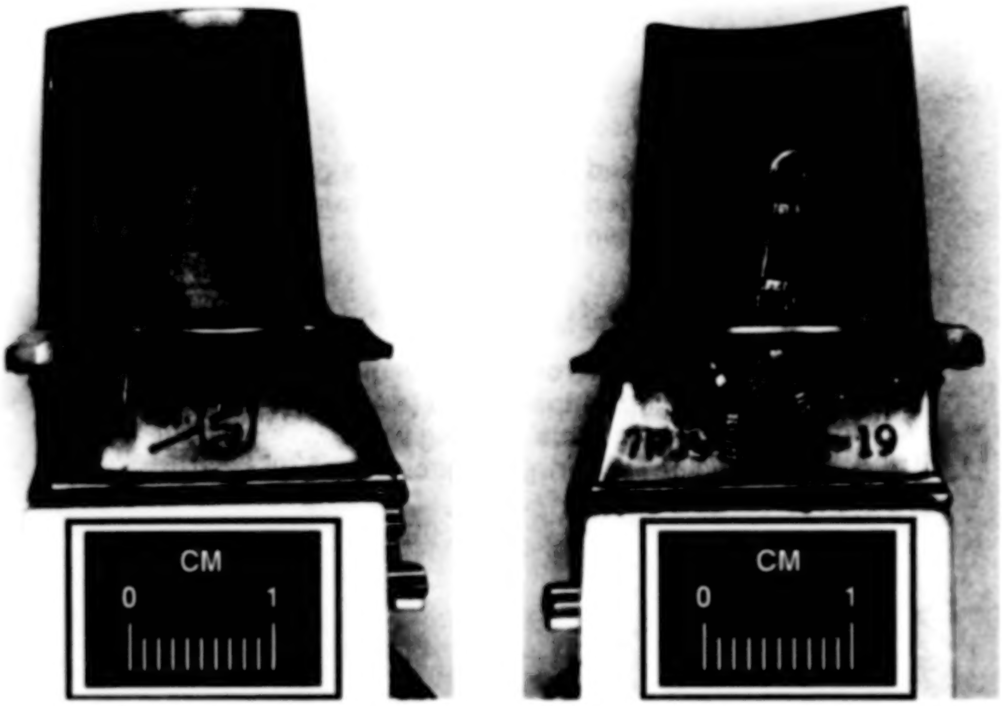
- Contact Sensors
  - Plug-type and thin-film heat flux gages
  - Thin-film thermocouples on ceramics
  - Surface strain gages

- Remote Optical Measurement Systems
  - Speckle interferometry for remote strain
  - Multipoint/multiparameter (NASP)
  - PC-based particle imaging velocimetry
  - Rayleigh scattering diagnostics

CD-91-54217

This report will cover a sampling of both the minimally intrusive contact sensors and the nonintrusive optical measurement systems. This is not a complete listing of the work we are doing, but it is a representative sampling of the most recent and pertinent work.

Plug-Type Heat Flux Gage



CD-91-54218

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

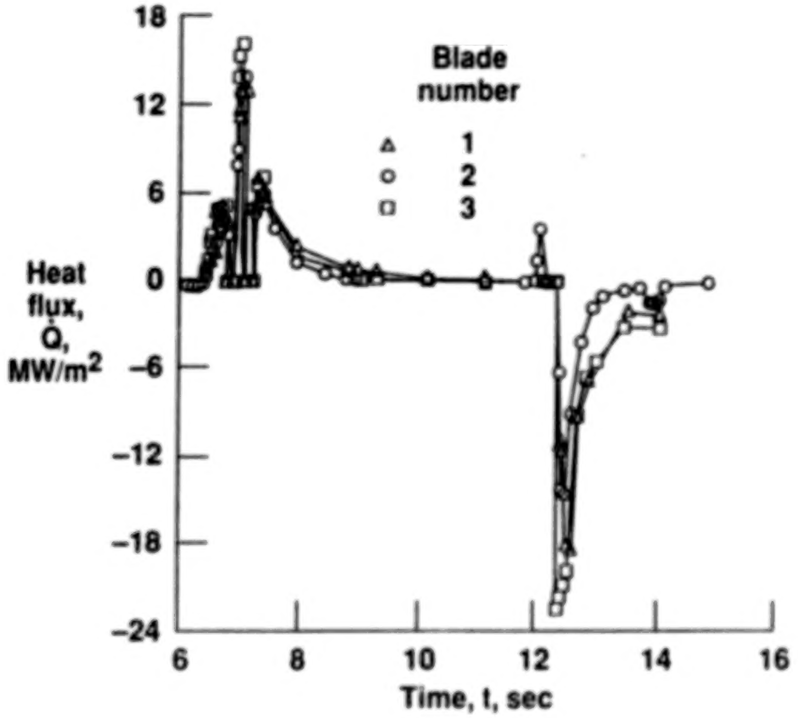
The first topic is heat flux gages. Our goal is to develop sensors to measure the heat flux on surfaces in the hot section, such as on blades or vanes.

The heat flux sensor pictured here is called a plug-type sensor. It is electron-discharge-machined right into the blade or vane and is therefore somewhat more intrusive to the blade than would be a thin-film sensor; however, it still doesn't disturb the gas flow.

This heat flux sensor is a post that is machined into the blade; the volume surrounding the post and behind it is filled with trapped air which acts as a thermal insulator. Very fine thermocouple pairs are splotted along the length of the post, and the wiring is brought out through small grooves machined into the backside of the blade. An additional thermocouple is buried in a small groove on the front side with the junction centered over the place where the post would be if it were machined all the way through. The post is treated as a one-dimensional heat flow device. The rates of change of the thermocouple signals can be processed to yield the instantaneous heat flux entering the front side of the blade.

The photographs show this gage installed on a space shuttle main engine (SSME) high-pressure fuel turbopump turbine blade. On the left is the front or suction side of the blade. This photograph was taken before a thin foil was welded over the thermocouple leads to smooth the installation. Similarly, the right photograph shows the back or pressure side prior to the closeout for smoothing. The rear of the post is visible in this figure.

Heat Flux Measured in SSME Turbine Blade Tester



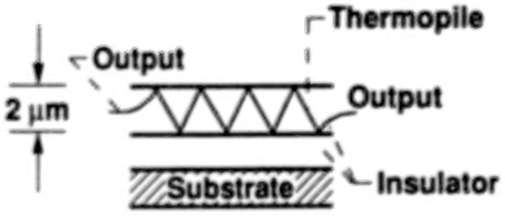
CD-91-54242

The blade in the previous figure (blade 2 in this graph) and two more just like it were tested in a blade cascade cyclic tester at NASA Marshall Space Flight Center. Each blade contained one of the plug type gages, but their locations varied.

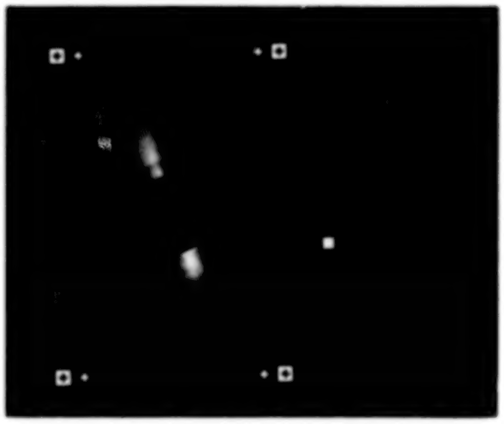
The tester produces thermal transient conditions very similar to those experienced in the startup and shutdown of the SSME. Blade 2 was mounted in the center of the three-blade cascade where the conditions were most like those in the turbine. Previous modeling results had shown that the transient heat flux was very large and potentially damaging to the blades. This had not been experimentally verified until this test.

This graph shows a plot of the heat flux in megawatts per square meter versus time in seconds for each of the three blades. The initial (positive) flux is a result of the cold blade being suddenly exposed to the combusting hydrogen and oxygen. The later (negative) values are a result of the blades being suddenly cooled by the cryogenic purge. Note that these heat flux levels are very large relative to those seen in aero turbines.

Thin-Film Heat Flux Gage



Principle of operation



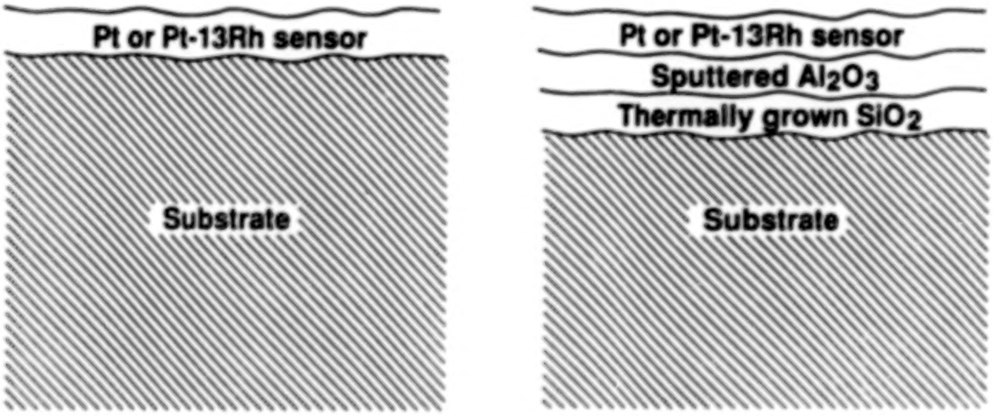
CD-91-54226

On the left in this figure is a schematic of our latest design of a thin-film heat flux sensor. The photograph on the right shows the most recent version of the design. This sensor is minimally intrusive to both the surface material and the gas stream.

The principle is to measure the temperature drop across a thin film of insulating material of known thermal conductivity. If the film is very thin, though, the temperature difference is so small that a single thermocouple pair cannot produce a signal large enough for reasonable measurement. Our approach is to capitalize on processes developed in the microelectronics industry to put a large number of thermocouple pairs on each side of the film and to connect them in series to form a thermopile. Thus, we obtain a usable signal from a very thin film.

The recent sensor pictured here has 100 thermocouple pairs on each side of a thin thermal barrier material. A few of these sensors have been fabricated, but work continues toward optimization of the many processing steps required. Success with this on flat surfaces will lead to the next step fabrication of this type of device on curved surfaces such as blades or vanes.

Thin-Film Thermocouples on Ceramic Substrates



Pt or Pt-13Rh sensor

Substrate

Pt or Pt-13Rh sensor

Sputtered Al_2O_3

Thermally grown SiO_2

Substrate

Electrically insulating ceramic substrates

Si_3N_4
Al_2O_3
Mullite

Electrically conducting ceramic substrates

SiC

CD-91-54222

At the last conference, in 1987, we showed the results of our program to develop thin, sputter-deposited thermocouples for metallic turbine blades and vanes. Such thermocouples are currently being applied and used routinely in development facilities.

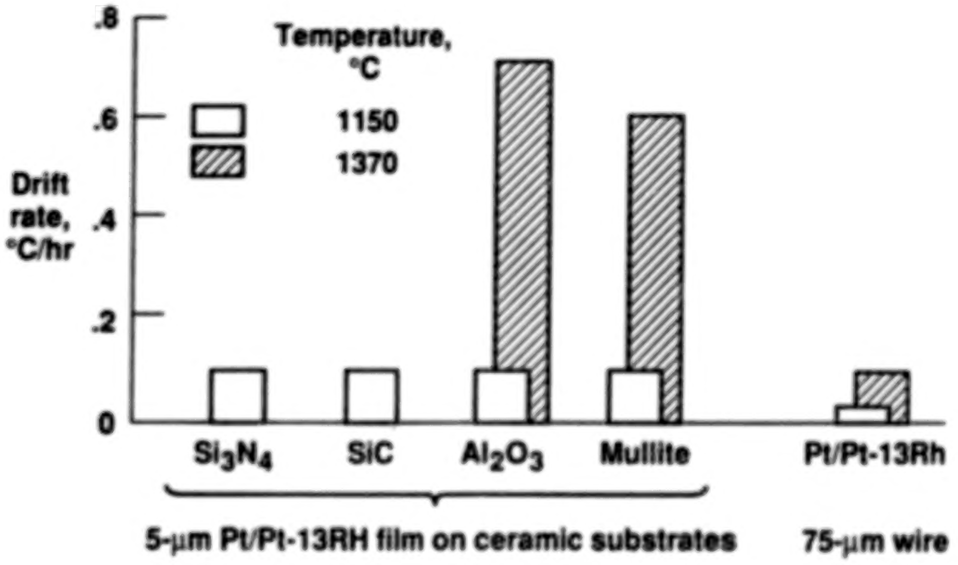
Our program is now driven by the need to make these surface temperature measurements at higher temperatures which typically means on ceramic materials. This figure illustrates the complexities of applying such sensors to different types of ceramics. Those which are electrically conducting require a carefully applied insulating layer under the thermocouple, as did the metals. Others, which are electrically insulating, do not require this extra step and thus are usually simpler.

Thin-Film Thermocouple on SiC Substrate at Two Stages of Fabrication Process

CD-91-54223

Here we see two thin-film, sputter-deposited Pt/Pt-13Rh thermocouples on SiC substrates. Two stages of the process are shown: the upper photograph shows the thermocouple before the lead wires have been welded to the films and the lower one shows a similar sample after lead wires have been attached.

Drift Rates for Thin-Film Thermocouples



CD-91-54224

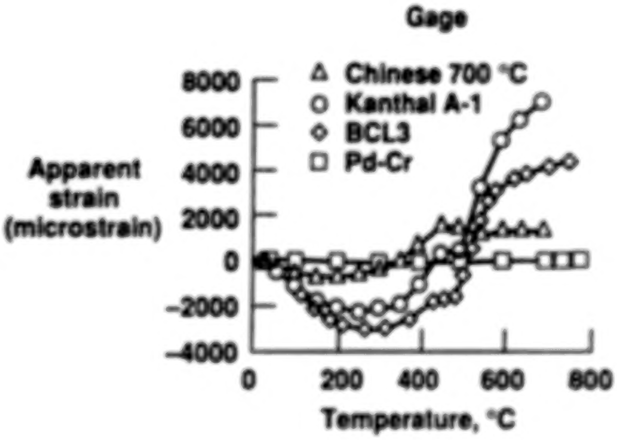
This chart shows the drift rate (in degrees Centigrade per hour) of thin film thermocouples on ceramic substrates held at elevated temperature. Data are shown for four different ceramic substrates at two temperatures. Drift data for comparable bare wire thermocouples are also included for comparison.

There are no data shown for Si_3N_4 or SiC at the higher temperature, because these materials exhibited an oxidation problem and a structural change, respectively, before reaching this temperature. This behavior makes thermocouples on these materials unusable at elevated temperatures.

Strain Measurement System Development

Miniature Resistance Strain Gage

Compensated Pd-Cr based wire strain gage



Apparent strain vs. Temperature
Pd-Cr gage factor, 1.3; others, 2.0

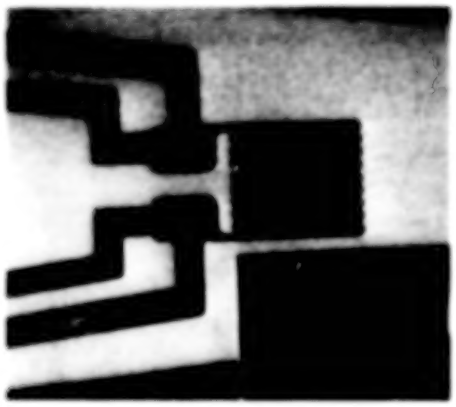
CD-91-54226

We are working on several approaches to measuring surface strain at high temperatures. The first extends the use of wire gages to elevated temperatures by carefully selecting the alloy. The second uses this same alloy in the form of a thin-film strain gages. The third approach uses a remote optical technique which will be discussed shortly.

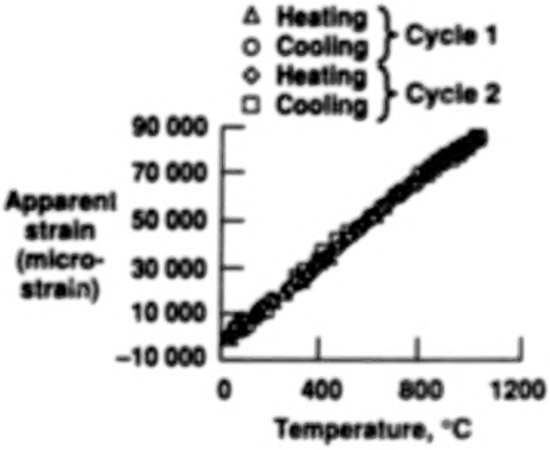
This figure shows wire gage made of a new alloy, Pd-Cr. Is temperature-compensated by use of a section of Pt resistance wire.

On the right, the residual apparent strain of this temperature-compensated Pd-Cr gage is compared with those of the best compensated gages previously available or reported. All of the latter gages are variations of the alloy Fe-Cr-Al. The much smaller residual apparent strain of the Pd-Cr gage demonstrates that compensation of this type of gage is far more successful than compensation of the Fe-Cr-Al gages.

Pd-Cr Based High-Temperature Static Strain Gage



Uncompensated Pd-Cr thin-film gage



Apparent strain vs. Temperature

Gage tested after thermal cycle to 1050 °C; gage factor = 1.8

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

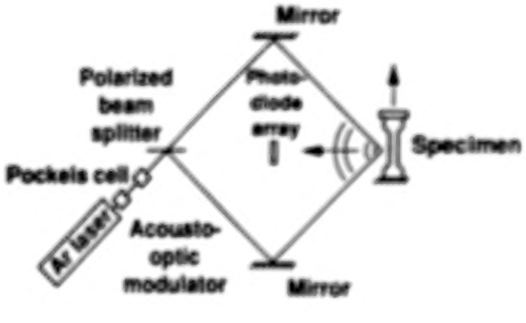
CD-91-54127

This figure shows the Pd-Cr alloy fabricated into a sputtered thin-film gage. This gage has not yet been temperature compensated as was the wire gage shown previously. We are currently working on the compensation.

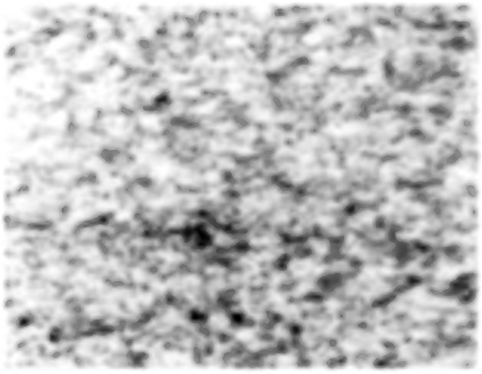
The plot on the right shows that the apparent strain versus temperature of this uncomensated gage is very repeatable over a wide temperature range and through multiple heating and cooling cycles. Repeatability of this nature is crucial if the gage is to be useable.

In addition, the plot is also very linear, which is a necessity for achieving minimal residual apparent strain after temperature compensation.

Speckle Interferometry for Remote Strain Measurements



Laser speckle strain measurement system



Speckle pattern

CD-91-54229

The remainder of this section will deal with the remote optical measurement systems.

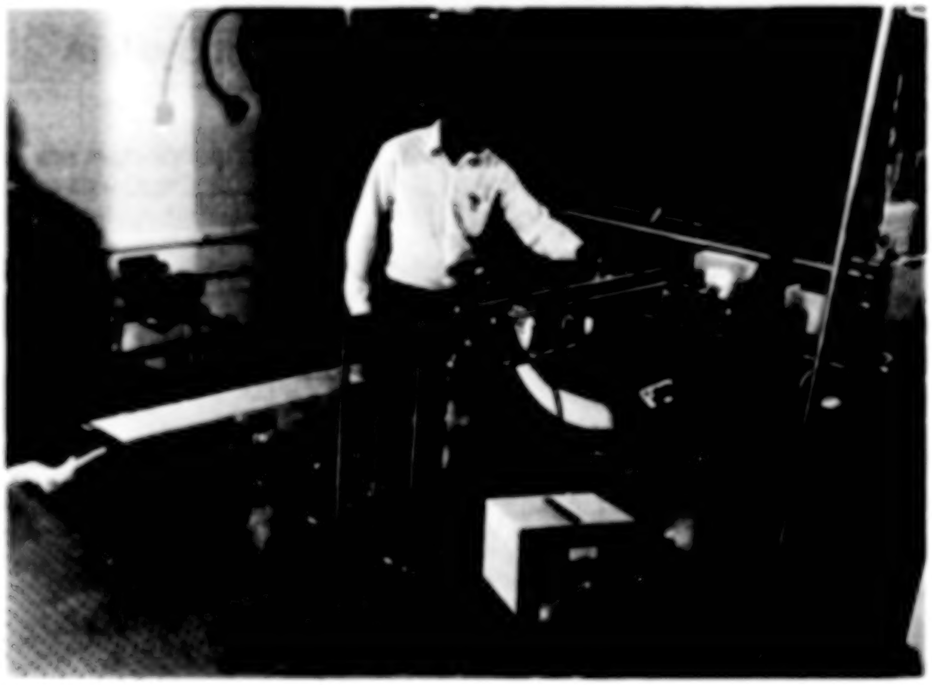
Here is shown an optical technique for remotely measuring strain. It is aimed at those situations where the temperature is beyond the gage alloys and/or where the surface strain is large enough to enter the plastic region.

The technique depends on the fact that the speckle pattern produced by impingement of laser light on a surface is caused by slight irregularities in the surface. This speckle pattern thus moves when the surface is strained. Electronic photographs of this pattern are taken before and after straining, and are then processed to track the speckles; thereby measuring the strain.

The left illustration shows the system schematically. The speckle pattern (example on right) is photographed before and after strain from two different directions. Processing of all four images allows the strain to be separated from any rigid body motion which might have occurred along with the strain.

A majora concern in this system is the large amount of processing required. Efforts are underway to reduce the processing time required.

Laser Speckle Strain Measurement System



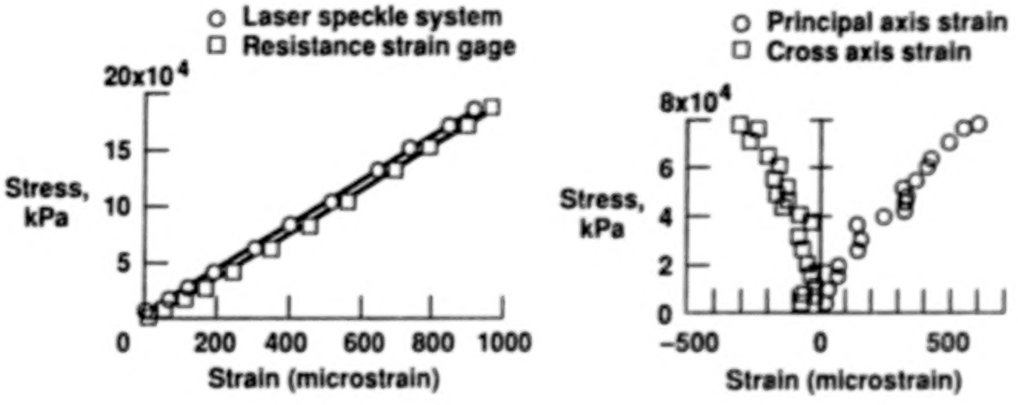
CD-81-54230

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This is a photograph of the laser speckle strain-measuring system in the laboratory. The tensile testing machine is on the right. This system was initially developed by using one-dimensional photodiode array, which simplified the processing relative to that required for a two-dimensional charge-coupled device (CCD) camera array. The center of this photograph shows the goniometer stage which allows the axis of the measurement to be varied so that two-dimensional maps can be generated.

Current work is concentrating on using a two-dimensional CCD camera and the attendant complexity of trying to process the strain map more rapidly. A second system is being developed for fiber applications.

Laser Speckle Strain Measurement System



Room temperature comparison with resistance strain gage

Measurement at 700 °C

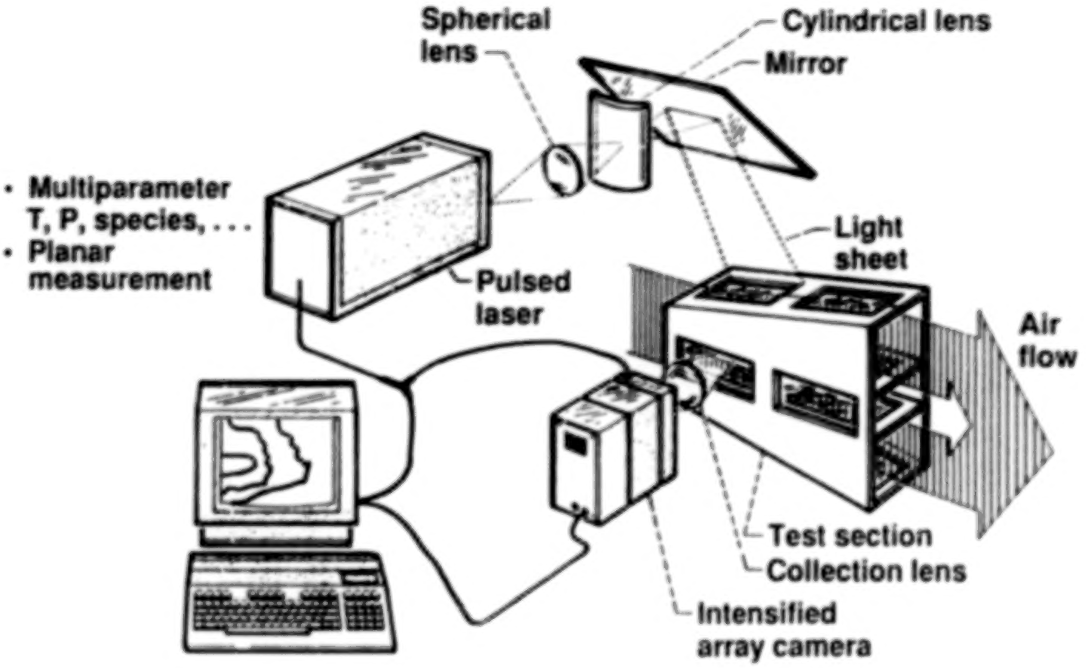
CD-91-54231

The plot on the left side of this figure shows a comparison of the strain measured by the speckle system with that measured by a conventional strain gage. These data were taken at room temperature so that the conventional strain gage could operate accurately. Agreement is excellent except for a small constant offset.

The other side of this figure shows data taken on a specimen held at 700 °C. The right half of this plot shows principal axis strain versus stress, and the left half shows cross axis strain versus stress. The slope of the principal axis data is in agreement with the elastic modulus for this material, and the slope of the cross axis data agrees well with Poisson's ratio.

Work is currently progressing toward testing specimens at higher temperatures.

Hypersonic Flow Diagnostic System


Spherical lens
Cylindrical lens
Mirror
• Multiparameter T, P, species, . . .
• Planar measurement
Pulsed laser
Light sheet
Air flow
Test section
Collection lens
Intensified array camera

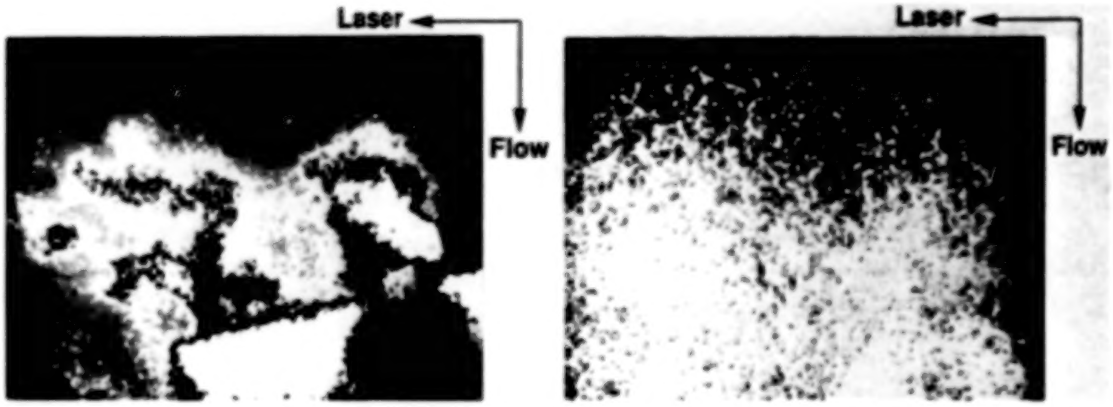
CD-91-54233

This is the Hypersonic Flow Diagnostic system currently being developed for NASP propulsion system research. The basic idea here is to illuminate a whole plane of the scramjet gas path with laser light and to use the resulting fluorescent emissions, spatially and spectrally resolved, to obtain a planar measurement of multiple variables. This includes such measurements as the concentrations of various species resulting from the combustion process and also the temperature of the gas.

The system is being extended by the use of two lasers and two cameras. This will allow two species to be measured simultaneously or temperature to be measured, by means of two spectral lines. Some major species, such as water, cannot be detected with this technique of laser-induced fluorescence. A Raman scattering based subsystem is being added to measure these species. However, Raman scattering is too weak to be reliably detected when the beam is spread into a sheet, so these will be point, not planar, measurements.

We plan to extend this to velocity measurement also.

Planar Laser-Induced Flourescence Measurements of OH Concentration in a Scramjet Combustor



Short

Long

CD-91-54234

ORIGINAL PAGE IS
OF POOR QUALITY

Shown here are the results of using a prototype of this system on the exhaust of a scramjet combustor. These represent the OH concentration for a short combustor configuration (left) and for a longer one (right). The more uniform mixing achieved with the longer combustor is evident in this picture.

Similar maps have been achieved for other species concentrations as well as for temperature distribution. Quantitative as well as qualitative data will bae obtained from this system.

Particle Imaging Velocimetry

Lewis Developed Vector Scanning Technique

Provides 2-dimensional map of velocity magnitude and unambiguous direction

Features

- Provides fast accurate data reduction without specialized processing hardware (hours of processing time reduced to a few minutes on a PC)
- All-video system for electronic processing
- Compatible with low number-density seeding
- Won R&D 100 award for 1989

Particle imaging velocimeter

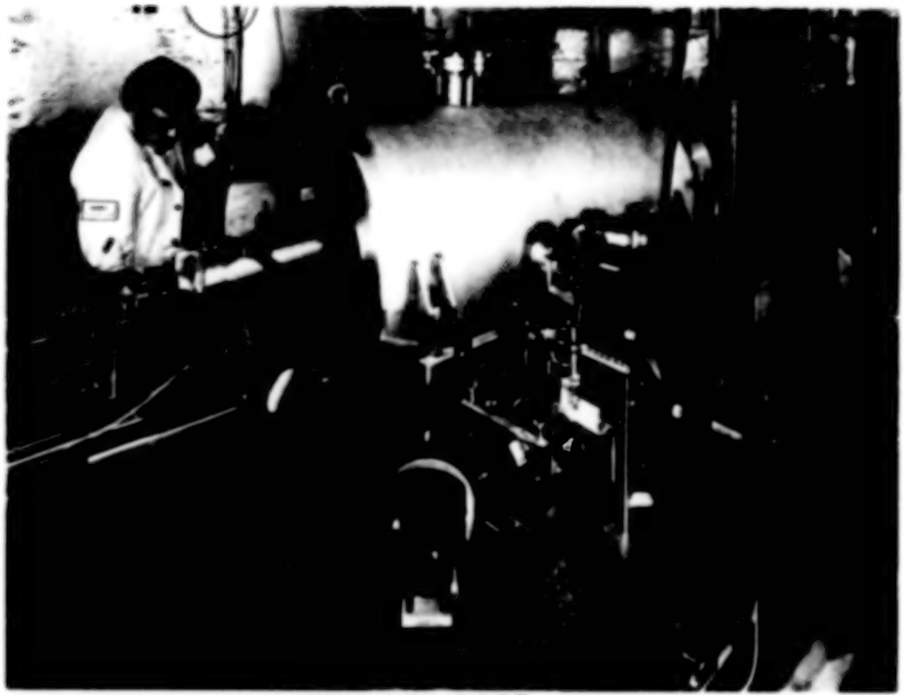
CD-91-54236

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The particle imaging velocimeter, shown on the right in this figure, is a method of mapping the flow velocity in the gas path. By repeatedly photographing (electronically) an entrained seed particle distribution as it progresses in time, one can identify the tracks of individual particles by means of image processing, and the velocity distribution can be measured.

The technique has been successfully applied to low velocity flows and is now being developed for flow velocities up to 100 m/sec.

The major features of this system are enumerated in the figure. I would like to emphasize that the data processing is quite rapid and requires only a PC-type computer.



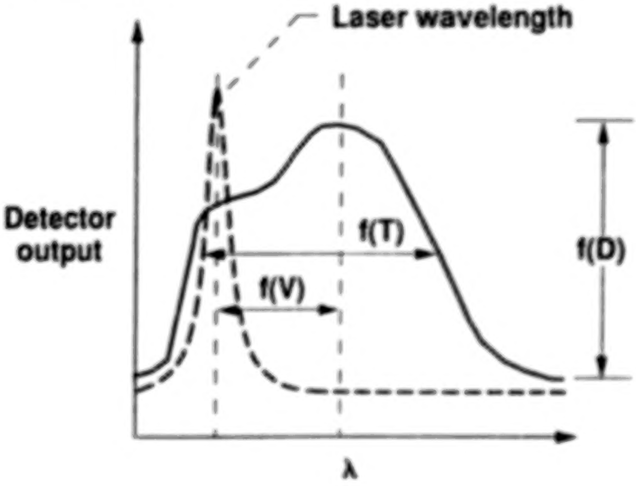
CD-91-54236

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

This is a photo of our newly developed Raleigh scattering system, which is used here to measure several parameters of the exhaust plume from a small hydrogen-oxygen rocket. A laser beam is focused at the point where the measurement is desired. The Rayleigh scattered photons are then collected and analyzed.

Rayleigh Scattering Diagnostics

Typical multiparameter data of plume velocity, V (4180 m/sec ± 4%), temperature, T (1110 K ± 3%), and density, D



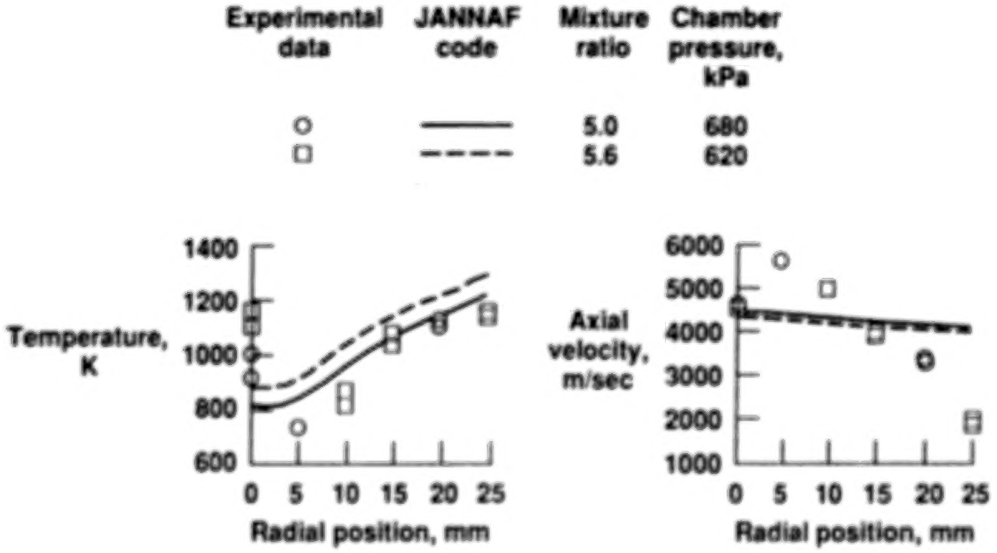
CD-91-54239

The interpretation of Rayleigh scattered light is dependent on the density regime.

For this rocket exhaust plume application, the parameters measured are (1) temperature, as determined from the line width of the received spectrum, (2) velocity, as determined by the displacement of the returned line from that of the incident laser, and (3) density, as determined by the amplitude of the returned line. This interpretation is valid only for situations of low density.

This technique is also being applied to temperature measurements in a 7-MN/m^2 hydrogen furnace. However, this high-density regime requires significantly different data interpretation.

Rayleigh Spectroscopy of Small H-O Rocket Plume



CD-91-54240

As stated previously, the goal of many of these measurement systems is the validation of computer codes. The plume data shown here were the first of their kind obtained. These data revealed that the code predictions were accurate in predicting the average values of temperature and velocity, but inadequate in describing the radial variations of these parameters.

What Will the Use of These Thin-Film Sensors and Optical Systems Require of You?

- **Investment in facilities, people, or closely coordinated contract, because the thin-film sensors must be fabricated on your specific hardware (not like buying a probe)**
- **Investment in highly skilled personnel to properly use the optical systems and to interpret the complex data that result from them**

"Owning a Stradivarius doesn't make one a violinist"

CD-91-54261

This report has dealt with a variety of contact sensors, primarily thin film, and a variety of remote optical measuring systems. Their development has allowed us to make measurements which previously could not have been made; however, they come at a price.

The thin-film sensors must be fabricated with fairly sophisticated equipment, such as sputtering systems, on the very part on which the measurement is needed. In addition, application to each new material requires some sensor redevelopment. As a result a significant investment in both equipment and skilled personnel is required to achieve these sensors.

Similarly, the remote optical systems are frequently quite complex, both optically and in the data reduction area. Packaged systems can sometimes be purchased, but they cannot be used effectively by any but highly skilled personnel.

"Owning a Stradivarius doesn't make one a violinist."

N91-20101

HIGH TEMPERATURE ELECTRONICS

Gary T. Seng
NASA Lewis Research Center
Cleveland, Ohio

In recent years, the aerospace propulsion and space power communities have acknowledged a growing need for electronic devices that are capable of sustained high-temperature operation. Aeropropulsion applications for high-temperature electronic devices include engine ground test instrumentation such as multiplexers, analog-to-digital converters, and telemetry systems capable of withstanding hot section engine temperatures in excess of 600 °C. Uncooled operation of control and condition monitoring systems in advanced supersonic aircraft would subject the electronics to temperatures in excess of 300 °C. Similarly, engine-mounted integrated electronic sensors could reach temperatures which exceed 500 °C.

In addition to aeronautics, there are many other areas that would benefit from the existence of high-temperature electronic devices. Space applications include power electronic devices for space platforms and satellites. Since power electronics require radiators to shed waste heat, electronic devices that operate at higher temperatures would allow a reduction in radiator size. Terrestrial applications include deep-well drilling instrumentation, high power electronics, and nuclear reactor instrumentation and control.

To meet the needs of the applications mentioned previously, the high-temperature electronics (HTE) program at the Lewis Research Center is developing silicon carbide (SiC) as a high-temperature semiconductor material. Research is focused on developing the crystal growth, growth modeling, characterization, and device fabrication technologies necessary to produce a family of SiC devices.

Interest in SiC has grown dramatically in recent years due to solid advances in the technology. Much research remains to be performed, but SiC appears ready to emerge as a useful semiconductor material.

High-Temperature Electronics Program

- Develop the base technology to establish silicon carbide as an advanced, high-temperature semiconductor material.
- Develop a family of silicon carbide integrated electronic sensors and other electronic devices for a variety of high-temperature and high power aerospace applications.
  - Need for high-temperature electronics, benefits of silicon carbide.
  - Growth of electronic-quality silicon carbide.
  - Device characteristics.
  - Focus of current and future research.

CD-91-53954

The high-temperature electronics (HTE) program is aimed at developing the base technology to establish silicon carbide as a high-temperature semiconductor material. Research is focused on developing the crystal growth, growth modeling, characterization, and device fabrication technologies necessary to produce a family of silicon carbide integrated electronic sensors and other electronic devices. Such devices would find numerous important applications in aerospace propulsion, space power, and high-temperature terrestrial systems.

The Need for High-Temperature Electronics

Commercial silicon devices are generally available for use up to 125 °C. Silicon technology is limited to a maximum temperature of 300 °C. There is an increasing need for electronics which reliably operate at a sustained temperature of 400 °C or greater.

Aerospace Propulsion Applications

- **Engine ground-test instrumentation.**
- **Control and condition monitoring systems.**
- **Integrated electronic sensors.**

CD-91-53955

On the basis of the inherent solid-state properties of silicon, the maximum temperature at which a silicon device could theoretically operate is 300 °C. Gallium arsenide could theoretically operate at 460 °C, but it is not stable at this temperature. However, there is an increasing need for electronics which operate at sustained temperatures at or above 400 °C. Engine ground-test instrumentation requires multiplexers, analog-to-digital (A/D) converters, and telemetry systems capable of withstanding hot-section temperatures in excess of 600 °C. Uncooled operation of control and condition monitoring systems in advanced supersonic aircraft would subject the electronics to temperatures in excess of 300 °C. Similarly, engine-mounted integrated electronic sensors could reach temperatures which exceed 500 °C. Additionally, for production aircraft applications, component reliability is of prime concern, and the imposed safety margin further increases the required temperature capability of any semiconductor selected for high temperature use.

Benefits of Silicon Carbide as Semiconductor Material

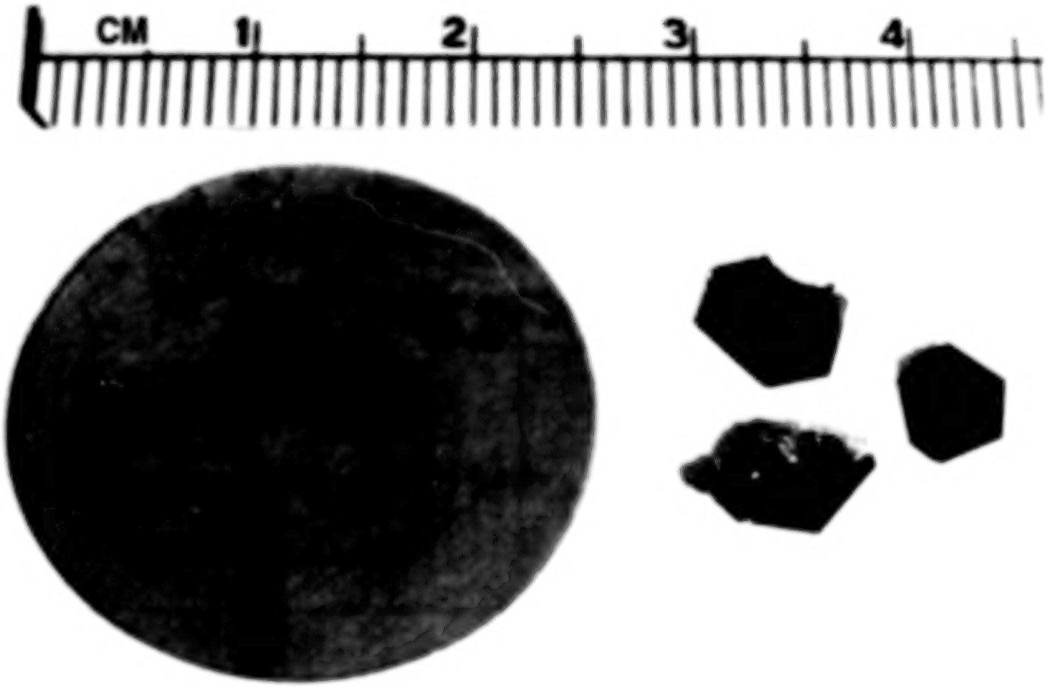
Property	Benefit
• Wide bandgap energy	• 400-600 °C electronics and radiation hardened devices
• Excellent stability	• Sustained use in hostile environments
• High breakdown field and high thermal conductivity	• Improved power electronics and increased device packing density
• Excellent high-frequency properties	• Superior high-frequency devices
• Processing similarities to Si	• Potential for rapid commercial development

CD-91-53856

With the same criteria as applied to silicon, silicon carbide could theoretically be employed at temperatures as high as 1200 °C. More reasonable, shorter term goals are to produce electronics capable of first 400 °C, then 600 °C operation. Current silicon carbide (SiC) semiconductor technology is reaching a level where production 400 °C electronics could be achieved in the near-term, while 600 °C electronics will be more of a challenge.

Because of the wide bandgap, and high operating temperature, SiC devices promise to be superior in radiation hardness over those using silicon or gallium arsenide. This material is also characterized by excellent physical and chemical stability, which make it suitable for long-term use in high-temperature, corrosive environments. The combination of the material's high breakdown field and high thermal conductivity provides the potential for improved power system electronics and for increasing the number of devices per unit area. Those properties, which determine the high-frequency characteristics of semiconductors, appear to be excellent for silicon carbide and superior to those of silicon or gallium arsenide. Finally, some similarities to well-established silicon processing techniques exist for SiC and this should aid in promoting its commercial development.

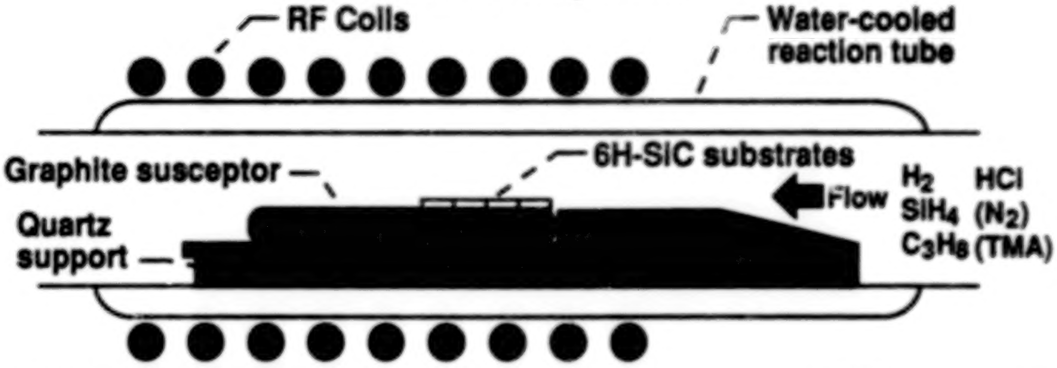
Silicon Carbide Crystals



Wafer (Cree Research, Inc.) **Lely crystals** CD-91-53987

Prototype silicon carbide devices were demonstrated above 400 °C as early as 25 years ago, but most research in the U.S. was terminated in the early 1970's because of an inability to produce the large-area, high-quality crystals required for commercial production. Unlike silicon and gallium arsenide, which are grown from a melt, silicon carbide (SiC) has no liquid phase and must be grown from the vapor state by sublimation. As is shown by the Lely process crystals in the photograph, the quality and size of SiC crystals were extremely difficult to control. However, in 1982, researchers at NASA Lewis developed a chemical vapor deposition (CVD) process which permitted the growth of thin, epitaxial films of SiC on standard silicon semiconductor wafers. For this discovery, an I-R 100 award was presented. Through the 1980's, several groups worldwide employed this method to grow SiC, and the work continues today. However, a recent development in SiC crystal growth is having an enormous impact on the research. A SiC research team at North Carolina State University announced the successful implementation of a seeded-growth sublimation method to produce SiC in boule (large cylinder) form. A private company, Cree Research Inc., has developed this process to the point where 1-in. diameter wafers of SiC are now being produced. Westinghouse Electric Corp. has duplicated this capability. The significance of this accomplishment is that SiC crystals can now be used as substrates for SiC expitaxial growth via the CVD processes already developed. The growth of very high quality SiC epitaxial films has now been achieved at NASA Lewis using these substrates.

SiC Chemical Vapor Deposition Reaction System



CD-91-53958

A diagram of the chemical vapor deposition (CVD) reaction system is presented. A silicon or silicon carbide substrate is placed on a radiofrequency-heated graphite susceptor. Rapidly ramping the temperature of the silicon (Si) substrate in the presence of silane (SiH_4) and propane (C_3H_8) in a hydrogen carrier gas produces single-crystal SiC. Depending on the substrate surface preparation employed, and the growth conditions, various crystalline forms of silicon carbide, called polytypes, can be grown. Doping (intentional insertion of the extrinsic carriers) the epitaxial films with electrical impurities to produce *n*-type and *p*-type SiC is vital to the realization of electronic devices. Addition of nitrogen gas to the growth process gases results in nitrogen incorporation into the SiC lattice. Since nitrogen is a donor impurity in SiC, *n*-type SiC is produced. To produce *p*-type SiC, aluminum has been used as an acceptor impurity. Aluminum is incorporated by adding trimethylaluminum to the growth process gases.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Chemical Vapor Deposition System

CD-91-53959

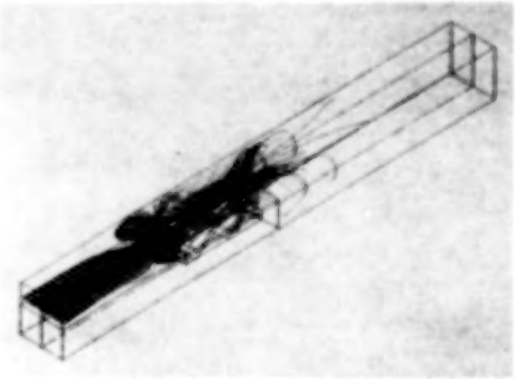
A photograph of the chemical vapor deposition system is shown. The reaction tube and associated plumbing are in the hood (left). The control valves and flow controllers reside in the vented area directly to the right. In the right foreground are the manual and computer control systems. The radiofrequency generator is not shown.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Chemical Vapor Deposition System Modeling

Objective: Provide an understanding of the effects of transport phenomena and chemistry on a global scale to improve SiC film quality

Modeling station



CVD flow modeling

Results/current research

- An increase in high quality SiC film growth area resulted from modifying the system susceptor as indicated by flow modeling studies
- Current study results will be used to guide CVD reactor scale-up to full wafer growth capability

CD-91-53960

While much empirical data has been generated for the CVD process, it is not well understood from a fundamental scientific standpoint. It is a very complex technique which can be difficult to experimentally optimize and particularly difficult if one is faced with predicting the effects of needed system modifications. As a result, the HTE program has vigorously pursued studies in CVD reactor modeling. Current research involves attempts to model both transport phenomena and chemistry in the reaction chamber. The results to date have provided solid guidance on system improvements and, we believe, hold much promise for the future in advancing SiC CVD growth technology.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Silicon Carbide Crystal Characterization and Processing

Crystal characterization techniques

Crystal processing techniques

CD-91 53961

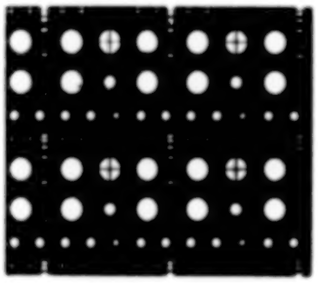
In addition to SiC crystal growth and growth modeling, a variety of crystal characterization and processing techniques are employed and studied in the HTE program. Electrical, chemical, and morphological characterization of samples is key to evaluating progress in the area and guides crystal processing and growth efforts. Selected examples of facilities are shown here. Clockwise from top left they are optical microscopy, scanning electron microscopy, oxidation, and crystal dicing. The Lewis program also sponsors several excellent characterization efforts at Case Western Reserve University, the University of Pittsburgh, and Howard University.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

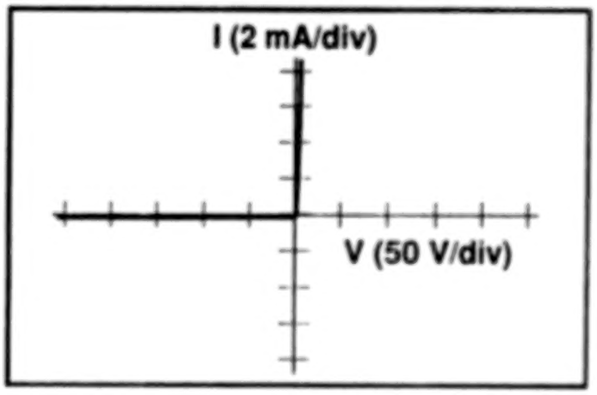
Silicon Carbide Junction Diode

Accomplishment: Highest reported operational temperature (600 °C) for any p-n junction diode device. Significantly improved characteristics above 400 °C. Demonstrates high quality 6H-SiC epitaxial film growth processes.

Diode array



I-V Characteristics at 600 °C



Benefits: Silicon carbide diodes (p-n junctions) are basic building blocks from which all future silicon carbide electronic devices will be developed

CD-91-53962

The availability of SiC substrates, combined with progress in processing and growth, has permitted NASA Lewis to place increasing emphasis on device fabrication. In-house research is pursuing the fabrication of grown junction diodes, metal-insulator-semiconductor field effect transistors (MISFET's), and metal-oxide-semiconductor field-effect transistors (MOSFET's). Recent results have been dramatic.

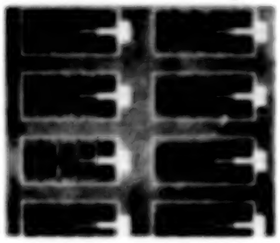
The first high-temperature prototype SiC device fabricated was a *p-n* junction diode. The junction structure was produced by first growing a nitrogen-doped *n*-type 6H-SiC film followed by an aluminum-doped *p*-type film. Photolithography and reactive ion etching were used to fabricate an array of mesa diodes. After passivating the junction boundary with silicon dioxide, metal contacts were then applied to the *p* and *n* regions.

A typical current-voltage (*I-V*) curve for one of the SiC grown-junction diodes is shown above. The function of a diode is to allow current to pass in one direction (the forward direction), but not in the opposite (reverse) direction. Hence, an ideal *I-V* curve would be nearly vertical in the forward direction, and nearly horizontal in the reverse direction. As is seen in the *I-V* curve, the diode exhibits excellent characteristics at 600 °C, the highest temperature measured. This is a significant result because the diode (i.e., *p-n* junction) is a basic building block for electronic devices.

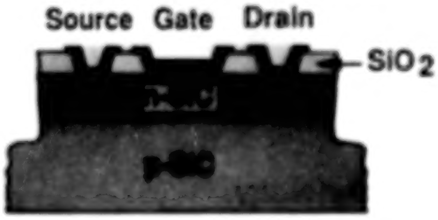
Silicon Carbide MOSFET

Accomplishment: A depletion-mode silicon carbide MOSFET has been developed and successfully demonstrated at an operational temperature of 500 °C.

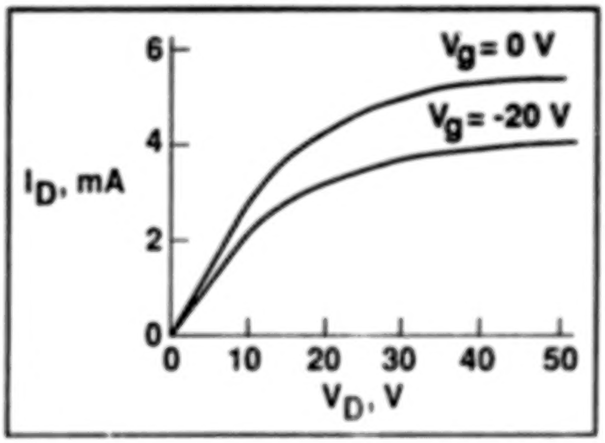
MOSFET array



SiC MOSFET structure



I-V Characteristics at 500 °C



Benefits: Silicon carbide MOSFET's (switches) provide the most basic active electronic device from which integrated circuits can be developed.

CD-91-53963

Another prototype SiC device fabricated at NASA Lewis was a metal-oxide-semiconductor field-effect-transistor (MOSFET). The MOSFET structure is shown above. For this device, the sequence of SiC films grown on the SiC substrate consisted of (1) an *n*-type buffer layer (not shown), (2) a *p*-type isolation layer, and (3) a 0.7-μm-thick *n*-type channel layer. An array of MOSFET's were then fabricated from this structure using many additional steps that included photolithography, oxidation, and metallization.

In an FET structure, the current flow is controlled by applying a voltage to the gate electrode. For an *n*-channel FET, a negative voltage applied to the gate will deplete the channel of electrons and thus "pinch-off" the current flow. In this manner, an FET resembles a switch. The switch is turned on and off by the application of the gate voltage. The *I-V* characteristics of the MOSFET at 500 °C are presented below. Although the electrical characteristics of the FET are not ideal, the achievement of transistor *I-V* characteristics at 500 °C is an extremely important starting point, since the FET provides the most basic active electronic device that can be employed to develop integrated circuits.

Focus of Current and Future Lewis High-Temperature Electronics Research

Current:

- Fundamental studies of SiC properties.
- CVD research to improve crystal quality.
- Detailed crystal characterization studies.
- CVD modeling studies.
- Device metallization research.
- Discrete device fabrication research.

Future:

- Continue all current research efforts.
- Perform research in passivation, wire attachment, and packaging.
- Demonstrate simple integrated circuit technology.
- Demonstrate integrated electronic sensor technology.

CD-91-53[illegible]

Current and future HTE research areas are outlined. The program scope is quite diverse and addresses a wide range of fundamental and applied areas critical to the future success of SiC. The HTE Group is heavily involved in a growing network of government, industrial, and academic programs in SiC. We will continue to participate in collaborative efforts to advance this technology as rapidly as is possible.

Future research will expand into more applied areas which will be addressed through a combination of in-house, collaborative, and sponsored research.

CONCLUDING REMARKS

Ultimately, the goal of the NASA Lewis High Temperature Electronics Program is to develop SiC integrated circuits and monolithic sensors with compensating and signal conditioning electronics integrated into the sensor structure. Electronic devices and sensors that are capable of operating at elevated temperatures eliminate or substantially reduce the amount of cooling that is required. Other payoffs of high temperature integrated sensors include reduced cabling and shielding requirements and the development of distributed control architectures, e.g., smart actuators.

Silicon carbide semiconductor technology was just a promise for many years, but has accelerated rapidly over the last several years. The devices fabricated at NASA Lewis and elsewhere demonstrate that excellent performance can be achieved. We want to emphasize that much development and optimization remain. However, the payoffs for success are tremendous. We believe that SiC can be an enabling electronic technology for many of the ambitious plans that mankind has for the Earth, air, and space.

N91-20102

FIBER-OPTIC-BASED CONTROLS

Gary T. Seng
NASA Lewis Research Center
Cleveland, Ohio

The challenge of those involved in control-system hardware development is to accommodate an ever-increasing complexity in aircraft control, while limiting the size and weight of the components and improving system reliability. A technology that displays promise towards this end is the area of fiber optics. The primary advantages of employing optical fibers, passive optical sensors, and optically controlled actuators over conventional control systems are greater immunity from electromagnetic effects, weight and volume reduction, non-radiating characteristics, superior bandwidth capabilities, and freedom from short circuits and sparking contacts. Since 1975, NASA Lewis has been performing in-house, contract, and grant research in fiber-optic sensors, high-temperature electro-optic switches, and "fly-by-light" control-system architecture. Passive optical sensor development is an essential yet challenging area of work and has therefore received much attention during this period.

A major effort to develop and demonstrate fly-by-light control-system technology, known as the fiber-optic control system integration (FOCSI) program, was initiated in 1985. Phase I of FOCSI, a NASA-DOD effort completed in 1986, was aimed at the design of a fiber-optic propulsion/flight control system. Phase II, a NASA-Navy effort currently in progress, will provide the system design, subcomponent and system development, and system ground tests. Phase III, flight demonstration, has also been initiated and will culminate in full FOCSI system flight tests. In addition to a summary of the benefits of fiber optics, the FOCSI program, sensor advances, and future directions in the NASA Lewis program will be discussed.

Fiber-Optic-Based Controls

- **Develop the technology neccesary to incorporate fiber-optic-based control systems into advanced aerospace vehicles.**
  - **Benefits of using fiber optics**
  - **Fiber-optic control system integration program**
  - **Fiber-optic sensors research**
  - **Future directions in fiber-optic-based controls**

CD-91-53843

Research in fiber-optic-based controls is aimed at developing the technology necessary to incorporate fiber-optic propulsion/flight control systems (fly-by-light) into advanced aircraft. The program includes the development and testing of passive optical sensors and electro-optic components, and the design, development and flight demonstration of a fiber-optic-based control system. An outline of the presentation is included.

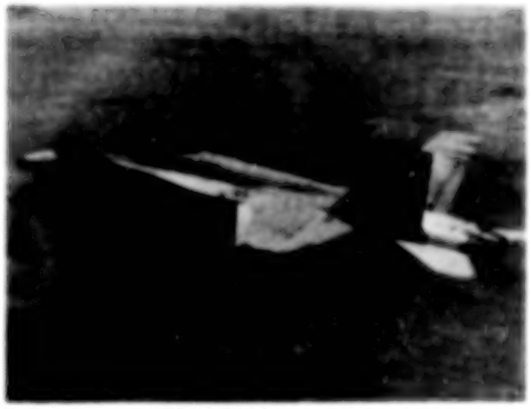
Benefits of Fly-By-Light Control Systems

- **Immunity from electromagnetic effects**
- **Weight and volume reduction**
- **Nonradiating**
- **High bandwidth**
- **Freedom from short circuits/sparking contacts**

CD-91-53944

The major benefits of a fly-by-light aircraft control system are outlined. Because optical fibers are dielectric, problems with electromagnetic effects (EME - electromagnetic interference, electromagnetic pulse, and lightning) are eliminated. This, in turn, eliminates the need for shielding and surge-quenching circuits. It is expected that replacing control-system electrical wiring with optical fibers will result in weight and volume savings, as well. Fiber-optic systems are nonradiating, and therefore leave no EM signature. The high bandwidth capability is advantageous for bus lines and offers the potential for all avionics data to be transmitted over a single line. The use of fiber optics also eliminates the threat of fires due to insulation failures or short circuits, which could also cause inadvertent actuation of control hardware.

Fiber Optic Control System Integration (FOCSI)



FOCSI Aircraft

Objective:
NASA–Navy program to develop and demonstrate a fiber optic (fly-by-light) propulsion/flight control system for advanced aircraft.

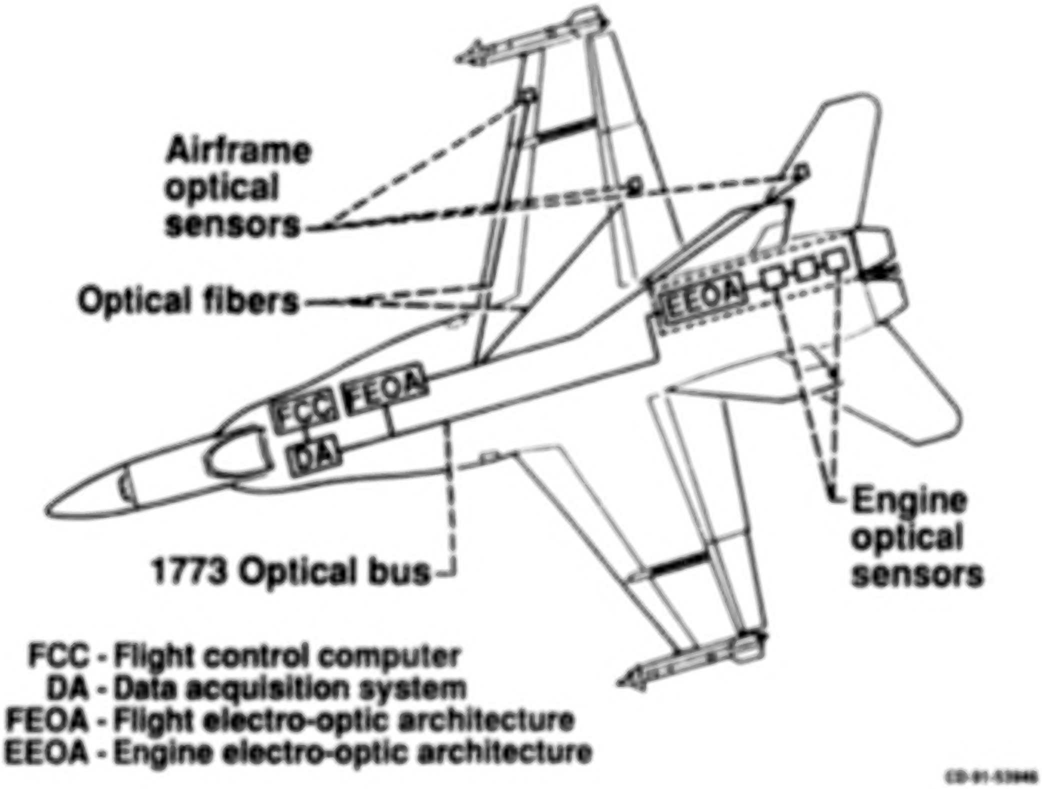
Program status:

- **Contracts for hardware build/environmental testing are on-going with McDonnell-Douglas and General Electric (awarded FY '89/'90).**
- **Flight tests of individual sensor systems on F-15 (NASA DFRF) to be initiated mid-FY '91.**
- **Flight tests of full FOCSI system to begin mid-FY'93 on F-18 (NASA DFRF).**

CD-91-53945

The design, development, and flight demonstration of a fiber-optic integrated propulsion/flight control system for an advanced supersonic fighter is the goal of the joint NASA and Navy FOCSI program. Phase I, to assess current technology and to provide system design options, was completed early in FY '87. Phase II, to design, fabricate, environmentally test, and ground test (engine, iron bird) the FOCSI system was initiated in mid-FY '88. Contracts for the hardware build and environmental testing of the sensor systems and electro-optic architecture were awarded to McDonnell-Douglas in FY '89 and General Electric in FY '90. All ground testing of the propulsion and flight fiber-optic-based systems will be completed early in FY '93. Phase III, flight demonstration, has also been initiated and will result in flight tests of individual sensor systems in FY '91 on an F-15 at NASA DFRF. The program will culminate with full FOCSI system flight tests to begin by mid-FY '93 on an F-18 at NASA DFRF.

Fiber Optic Control System Integration (FOCSI)

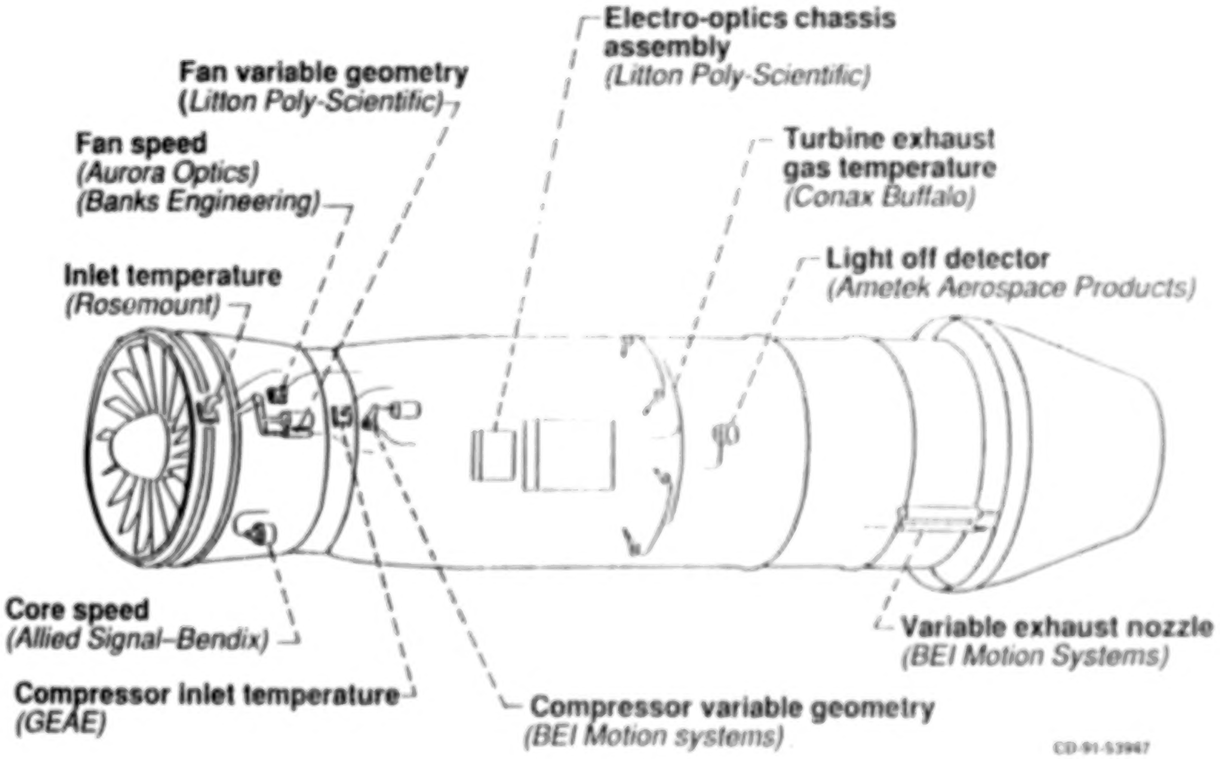

Airframe optical sensors
Optical fibers
FCC
FEOA
DA
EEOA
Engine optical sensors
1773 Optical bus
CD-91-53946

FCC - Flight control computer
DA - Data acquisition system
FEOA - Flight electro-optic architecture
EEOA - Engine electro-optic architecture

The overall FOCSI system configuration is presented. Propulsion and flight passive optical sensors will be linked to their respective electro-optic architectures (EOA's) via fiber optic cables. The propulsion and flight EOA's will communicate with the data acquisition system using a mil spec 1773 optical data bus. Within the data acquisition system, the conditioned optical sensor signals will be recorded and compared with the corresponding bill-of-material sensor signals.

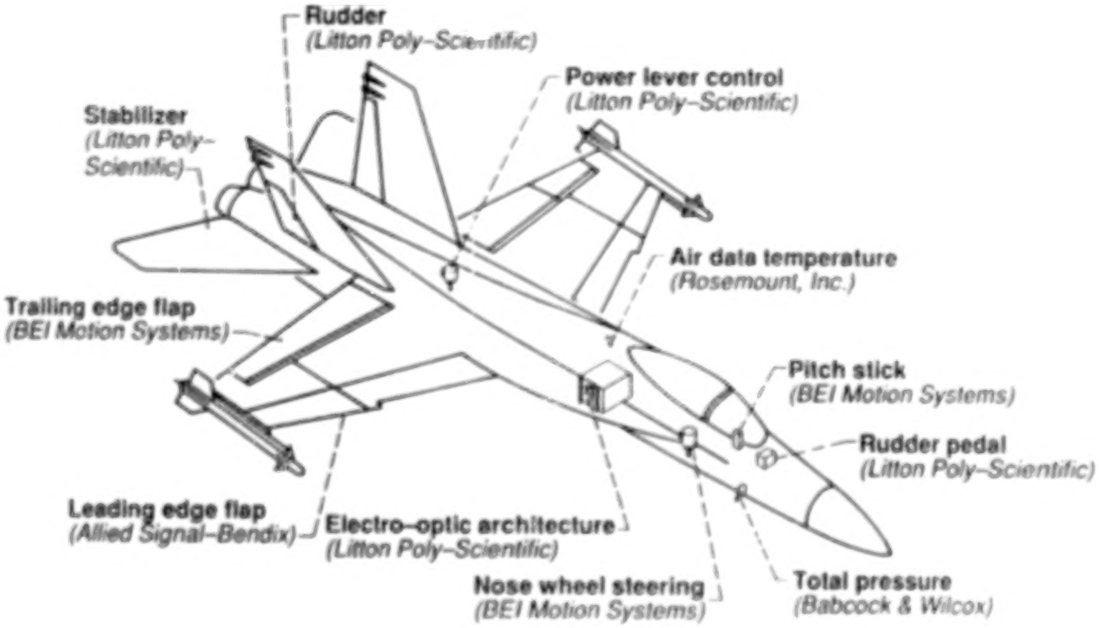
FOCSI Propulsion System Configuration

Propulsion System Prime–General Electric Aircraft Engines


Electro-optics chassis assembly
(Litton Poly-Scientific)
Fan variable geometry
(Litton Poly-Scientific)
Fan speed
(Aurora Optics)
(Banks Engineering)
Turbine exhaust gas temperature
(Conax Buffalo)
Inlet temperature
(Rosemount)
Light off detector
(Ametek Aerospace Products)
Core speed
(Allied Signal–Bendix)
Variable exhaust nozzle
(BEI Motion Systems)
Compressor inlet temperature
(GEAE)
Compressor variable geometry
(BEI Motion systems)
CD-91-53947

The full complement of FOCSI propulsion system sensors and the electro-optics chassis assembly (electro-optic architecture - EOA) are shown as they will be mounted on the engine. Subcontractors, selected by General Electric Aircraft Engines, the prime contractor, are listed below each sensor and the EOA. A major of emphasis of this program is to develop a strong vendor base of fiber optic sensor systems and EOA's.

FOSCI Flight System Configuration
Flight System Prime–McDonnell Aircraft Co.


Rudder
(Litton Poly-Scientific)
Power lever control
(Litton Poly-Scientific)
Stabilizer
(Litton Poly-Scientific)
Air data temperature
(Rosemount, Inc.)
Trailing edge flap
(BEI Motion Systems)
Pitch stick
(BEI Motion Systems)
Rudder pedal
(Litton Poly-Scientific)
Leading edge flap
(Allied Signal-Bendix)
Electro-optic architecture
(Litton Poly-Scientific)
Nose wheel steering
(BEI Motion Systems)
Total pressure
(Babcock & Wilcox)

CD-91-53948

As was previously shown for the propulsion system, the full complement of flight system sensors and the electro-optic architecture are presented. Subcontractors, selected by McDonnell Aircraft Co., the prime contractor, are listed below each sensor and the EOA.

Fiber Optic Sensor System Flight Tests

NASA DFRF F-15 (HIDEC)

Objective:
Lay a solid foundation for achieving FOCSI success by providing information on fiber optic sensor system installation, in-flight operation and maintenance

Status:

- **Flight tests (engine bay) of Bendix Faraday speed sensor and Lewis Research Center WDM position encoder to be initiated by mid–FY 91 on F–15 (HIDEC) aircraft.**
- **Vendors selected for T5 and T1 sensor flight tests (engine penetration) in FY 92 on F–15 (HIDEC) aircraft.**
- **Lewis Research Center thin film temperature sensor to fly (engine bay) in FY 92 test.**

CD-91-53949

Prior to installing and flying the full FOCSI system in FY '93, it was felt that select single sensor system flight tests would be worthwhile. This will permit early experience to be gained on sensor system installation, maintenance, and in-flight operation. Planned for this year are tests of a Faraday effect speed sensor developed by Allied Signal-Bendix to read engine compressor speed (N2) through the gearbox power takeoff pad, and a Lewis-John Carroll University wavelength division multiplexed optical position encoder which will measure throttle position (PLA). A second series of flights, slated for FY '92 will test a blackbody turbine discharge temperature sensor (T5) developed by Conax Buffalo and a fluorescence rate-of-decay inlet temperature sensor (T1) developed by Rosemount. Both sensors will penetrate the engine for these tests. Lewis will also supply a novel thin film temperature sensor for engine bay testing. All sensor system flight tests will be performed at NASA DFRF on the F-15 (HIDEC) aircraft. Work with DFRF in fiber optics was initiated in FY '88.

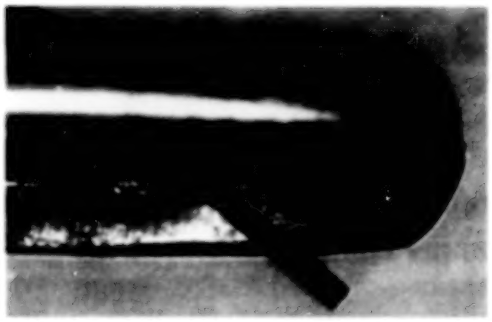
Fiber-Optic Sensors Program Accomplishments

• Tachometer demonstrated on engine	1976
• Position encoder demonstrated on compressor guide vane	1976
• Tip clearance sensor demonstrated on compressor stage	1980
• 800 °C temperature sensor developed	1980
• 1000 °C temperature sensor developed	1983
• Gallium arsenide photoswitch developed (260 °C operation)	1985
• High-temperature pressure sensor developed	1985
• 1700 °C gas temperature sensor developed	1986
• Improved intensity sensor referencing techniques developed	1987
• WDM position encoder demonstrated on engine	1988
• 1900 °C gas temperature sensor demonstrated on engine	1989
• Thin film temperature sensor developed	1990

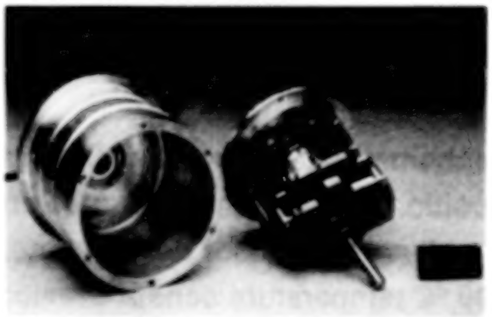
CD-91-53950

To implement a fiber-optic-based control system requires the development of passive (no electrical connections) optical sensors and optically controlled actuators capable of surviving aircraft environments. NASA Lewis has addressed this critical area of technology since 1975 by developing a wide variety of optical sensors and a high-temperature electro-optic switch through in-house, contract, and grant efforts. Major accomplishments for the fiber optic sensors program are presented for each year since its inception.

Fiber–Optic Sensors Research



Thin film temperature sensor



WDM position encoder

Intensity sensor referencing techniques

1900 °C blackbody temperature sensor (Conax Buffalo Corp.)

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-81-53951

Current in-house and sponsored research efforts vary from the development of near-term flight prototype control sensor systems through far-term investigation of innovative sensor/sensor system concepts and includes work in

- Flight Prototype aircraft control sensor systems
- On-engine demonstrations of sensor systems
- Laboratory demonstration/testing of new sensor concepts
- Improved sensor referencing /signal processing techniques
- Integrated optics/microfabrication techniques
- Electro-optic component research

The overall goal is to develop miniature, rugged, passive, optical sensor systems which operate reliably in the aerospace environment. Shown above are selected areas of current research, which include (clockwise from top left) a novel thin film temperature sensor, a wavelength division multiplexed optical encoder, a blackbody temperature sensor (SBIR-Conax Buffalo Corp.), and laboratory work to improve optical intensity sensor accuracy and precision.

Future Directions in Fiber-Optic-Based Controls

- **Continue efforts aimed at achieving FOCSI objectives.**
- **Continue development of novel fiber-optic sensor concepts.**
- **Engine test prototypes of promising fiber-optic sensor systems.**
- **Continue to sponsor research in the development of electro-optic components.**
- **Aid in achieving consensus on fiber-optic component specifications for aircraft.**

CD-91-53952

NASA intends to continue to aggressively pursue all the areas of fiber-optic technology presented. Future directions in the program are presented above. While much work in fiber-optic-based control systems remains, the technology appears well on its way to successful demonstration for aircraft.

One area which deserves more attention is fly-by-light control systems for future generation fighter aircraft and high-speed aircraft such as those flying in sustained supersonic or hypersonic regimes. Such control systems will require fiber-optic components capable of much higher temperature operation than was reported here.

Finally, integrated optics appears to be an area of technology that promises substantial benefits for fly-by-light systems by providing small, rugged, high-temperature, and inexpensive sensors. This is an area in which we intend to increase emphasis in the near future.

N91-20103

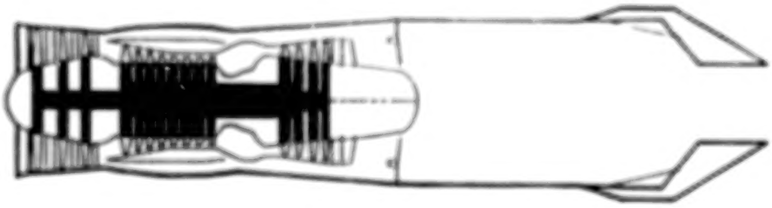
ADVANCED AEROPROPULSION CONTROLS TECHNOLOGY

Carl F. Lorenzo
NASA Lewis Research Center
Cleveland, Ohio

This paper discusses NASA Lewis' research activities in the area of propulsion control as driven by the trends and needs of advanced aircraft. Special emphasis is placed on research to develop design methodologies for integrated flight and propulsion control. The paper also covers research thrusts in hypersonic propulsion control and dynamics in support of the National Aerospace Plane, and a new concept for system critical component life-extending control is discussed.

Thrusts in Aeropropulsion Control

- Achieve the highest possible performance from the propulsion and aircraft systems
- Manage the dynamic coupling now being designed into advanced aircraft
- Minimize damage accumulation in system components
- Increase the availability of hardware through advanced control intelligence



CD-91-54246

The fundamental goals in aeropropulsion are toward lower weight, higher reliability, and higher performance systems. Improved performance takes the form of exotic special-purpose aircraft (such as supersonic short-takeoff, vertical-landing (STOVL) aircraft, the National Aerospace Plane (NASP), and X-Wing Convertible-Engine Rotorcraft). The challenge of advanced controls is to provide the intelligence and coordination so that the system components can realize the performance gains strived for at the component technology level and to maximize the reliability and utility of the new vehicles with their increased dynamic coupling between the vehicle and propulsion systems.

Project Focus Areas of Lewis Aeropropulsion Controls Program

- National Aerospace Plane—Propulsion system dynamic modeling and control
- Supersonic STOVL propulsion—flight control integration
- Life-Extending Control

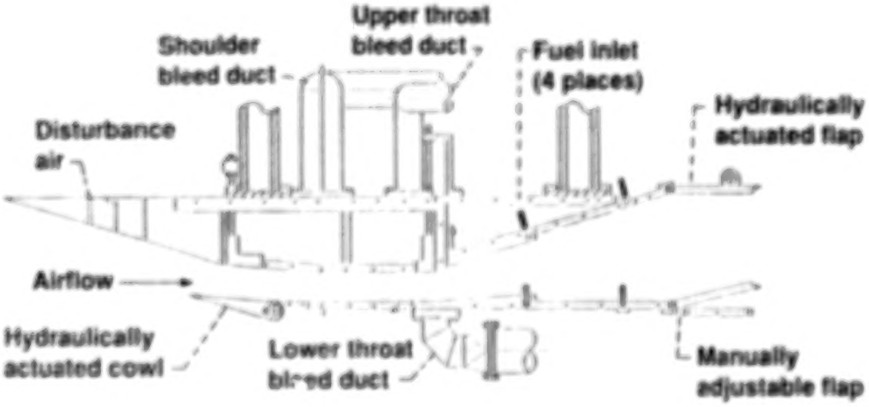
CD-91-54247

The Lewis Aeropropulsion Controls Program addresses the basic thrusts through focused studies of advanced aerospace vehicles and propulsion systems of broad interest.

NASP Propulsion System Dynamic Modeling

Objectives:

- Formulate dynamic models of NASP propulsion systems for control design and operability assurance
- Validate model through dynamic testing at selected operating conditions
- Determine transient and frequency responses, inlet unstart characteristics, and engine-to-engine interactions


Upper throat bleed duct
Shoulder bleed duct
Fuel inlet (4 places)
Hydraulically actuated flap
Disturbance air
Airflow
Hydraulically actuated cowl
Lower throat bleed duct
Manually adjustable flap

Modified Government Baseline Engine

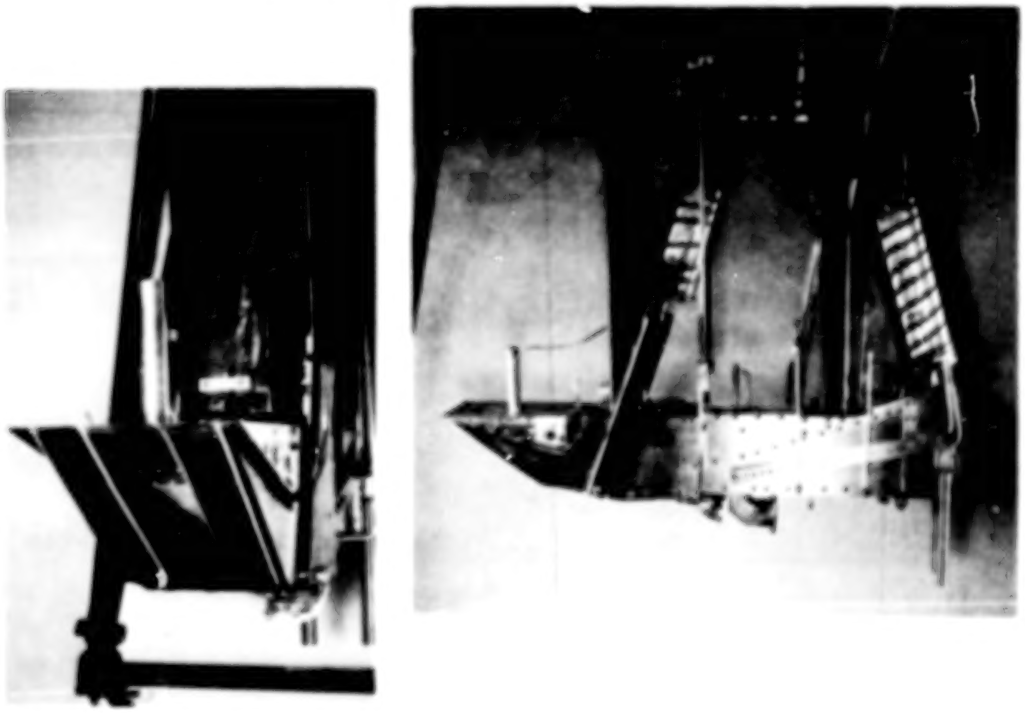
CD-91-54248

The NASP Propulsion System Dynamic Modeling Program seeks to create transient models of representative hypersonic propulsion systems. The intended uses of these models are to expose any broad operability problems and contribute to their resolution and to serve as a basis for hypersonic engine controller design.

The models that have been created are nonlinear, one-dimensional, transient codes that describe the overall behavior, including variable-geometry effects and combustion. These models are relatively simple to allow near-real-time operation for controls design but have sufficient fidelity to allow study of such problems as inlet unstart.

The Modified Government Baseline Engine (MGBE) is the focus of current modeling efforts.

Modified Government Baseline Engine

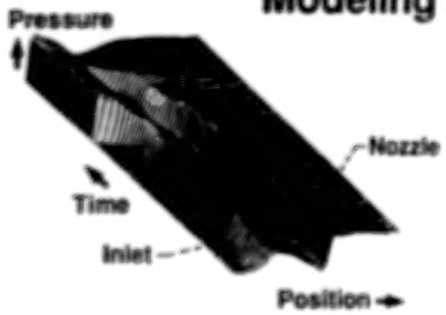


CD-81-54248

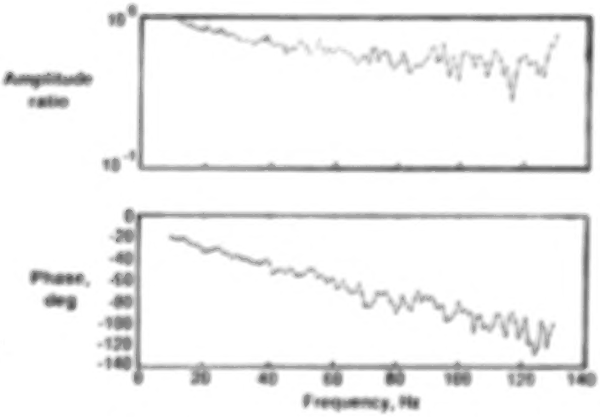
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The Modified Government Baseline Engine (MGBE), shown here, is being used to validate the dynamic model as well as the propulsion control design. The center duct is the active engine, and the two side ducts simulate adjacent engines. Their purpose is to allow engine-to-engine interaction to be studied.

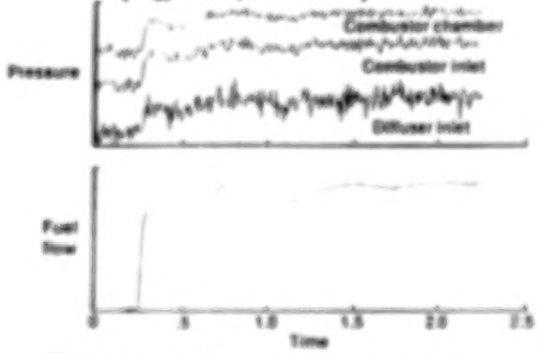
Hypersonic Propulsion System Dynamic Modeling and Test Results



Simulated engine unstart due to inlet Mach number perturbation ($M_\infty = 2.5$; $\Delta M = 2\%$)



Combustor-pressure-to-fuel-flow transfer function



Response of engine pressures to step in fuel flow

CD-91-54250

The figure shows a simulated engine unstart due to an inlet Mach number perturbation. At some time $t_0 + \Delta t$, a Mach number perturbation is introduced into the inlet of the one-dimensional simulation. The Mach wave reflects off the nozzle and travels back upstream, unstarting the engine.

The response of MGBE pressures (actual data) to a step in combustor fuel flow (calculated from manifold pressures) is also shown. At about 0.25 sec the fuel flow is stepped up. Shortly thereafter the engine pressures follow. Close inspection shows that the combustor inlet pressure rises first, followed by combustor chamber pressure and diffuser inlet pressure, respectively. The large low-frequency perturbation is the fuel system dynamics settling out after the request for increased flow.

A typical transfer function for engine combustor pressure to fuel flow is shown. A frequency sweep test signal was applied to the main fuel valve. The resulting transfer function shows that over the frequency range the engine exhibited a dead time of about 4 msec.

Hypersonic Propulsion System Control

Objectives:

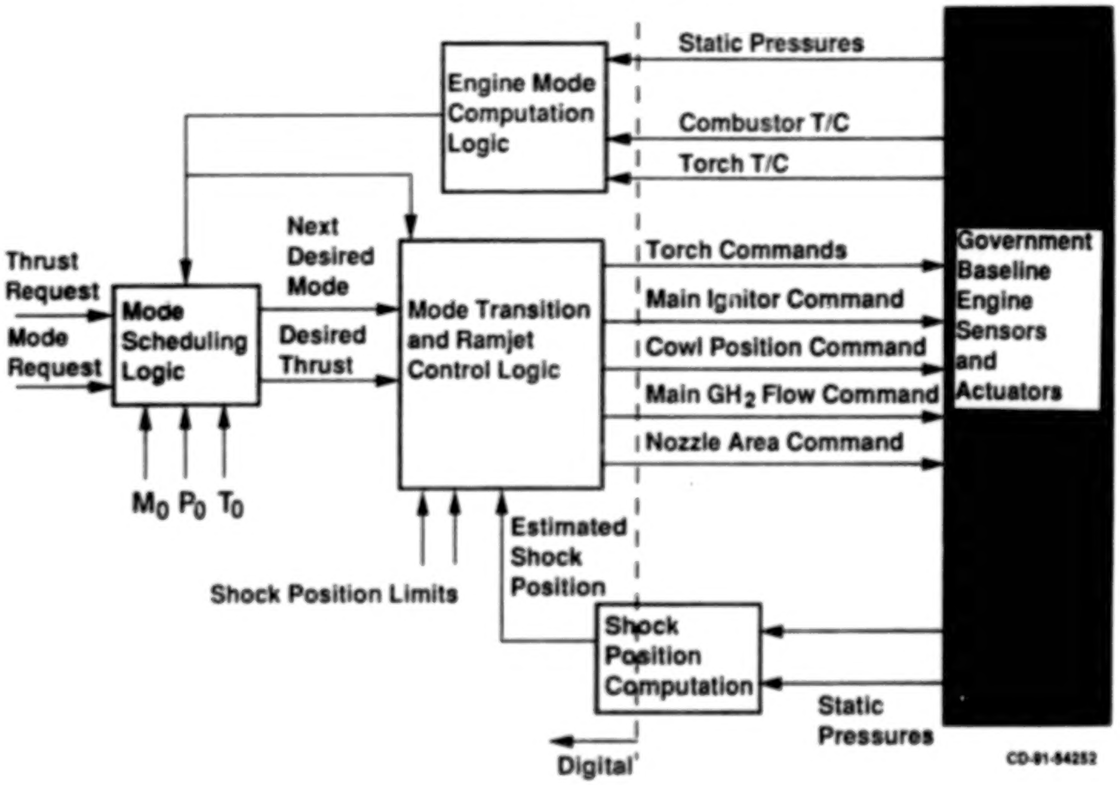
- Determine practical methods of control for hypersonic propulsion systems
- Evolve control design and design method for Modified Government Baseline Engine
- Validate through testing
- Demonstrate
  - Shock position management
  - Thrust control
  - Mode transition control (start/unstart/restart)

Inlet view of Modified Government Baseline Engine

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The thrust of this activity is to establish reasonable and practical control design approaches for hypersonic propulsion systems. The management of these complex multimode machines presents challenges in shock position management, thrust control, and mode transition control. Of particular importance is the issue of inlet unstart/restart management through coordinated control of engine fuel flow and variable geometry.

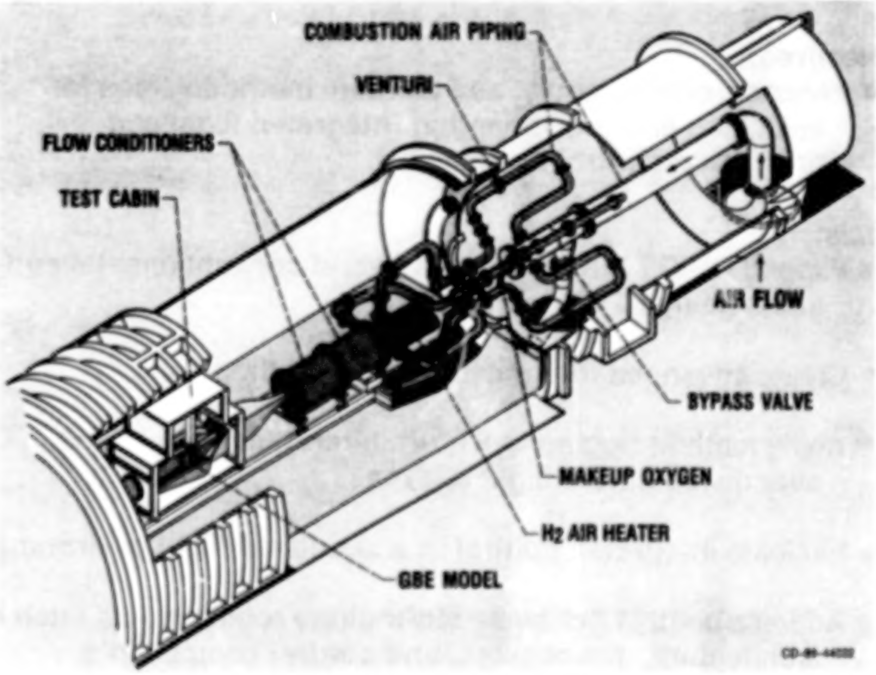
Modified Government Baseline Engine Control Logic Diagram


Static Pressures
Engine Mode Computation Logic
Combustor T/C
Torch T/C
Next Desired Mode
Thrust Request
Mode Request
Mode Scheduling Logic
Desired Thrust
Mode Transition and Ramjet Control Logic
Torch Commands
Main Ignitor Command
Cowl Position Command
Main GH₂ Flow Command
Nozzle Area Command
Government Baseline Engine Sensors and Actuators
M₀ P₀ T₀
Estimated Shock Position
Shock Position Limits
Shock Position Computation
Static Pressures
Digital
CD-91-54252

This figure is a block diagram of the engine control logic to be tested in the Lewis Propulsion System Laboratory (PSL-4). The logic performs four basic functions: (1) it computes the next desired engine mode and thrust level given a thrust and mode request; (2) it determines the present engine mode from sensor data; (3) it computes the location of the engine's internal shock train; and (4) it produces commands to the engine's valves and variable-geometry actuators. The control logic runs on microprocessor-based, special-purpose control computers. A series of PSL tests will be run to demonstrate the control's ability to rapidly modulate engine thrust and also to hold thrust constant in the face of externally imposed disturbances.

C-4

MGBE PSL–4 Installation


COMBUSTION AIR PIPING
VENTURI
FLOW CONDITIONERS
TEST CABIN
AIR FLOW
BYPASS VALVE
MAKEUP OXYGEN
H_2 AIR HEATER
GBE MODEL

CD-91-54253

The MGBE engine control will be validated by tests in the modified Propulsion System Laboratory facility shown here. The facility enables continuous free-jet testing of engines in this size class. A hydrogen-fueled preheater provides flow at appropriate temperature to a Mach 3.5 nozzle with makeup oxygen to allow proper engine combustion. Run time is limited by the thermal capacity of the engine model.

Flight and Propulsion Control Integration

Objectives:

- Enhance, create, apply, and validate methodologies for designing and implementing integrated flight and propulsion control systems

Goals:

- Expand DMICS methodology beyond conventional-takeoff-and-landing (CTOL) aircraft
- Create advanced integrated control methodologies
- Apply methodologies to STOVL aircraft configurations over the complete flight regime
- Evaluate integrated control in a simulated flight environment
- Address control hardware technology requirements such as architecture, processors, and control components

CD-91-54254

New and advanced aircraft concepts such as Supersonic STOVL, X-Wing rotorcraft, and hypersonic vehicles derive significant performance gains by allowing high levels of dynamic coupling not normally associated with conventional aircraft. The cost of this dynamic coupling is either increased pilot workload or a requirement for the control system to unify the behavior of the flight control and propulsion control systems. The goal of this effort is to create an integrated control design methodology that yields globally optimal performance with reasonable pilot workload.

Supersonic STOVL Flight and Propulsion Control Integration Program Elements

E7D/F110 STOVL integrated flight and propulsion control study:

- Apply DMICS methodology to STOVL aircraft
  - Ejector-augmented STOVL configuration (E7D/F110)
- Evaluate control concept with real engine and simulated aircraft in ground-based test
- Evaluate control concept and handling qualities on Ames VMS

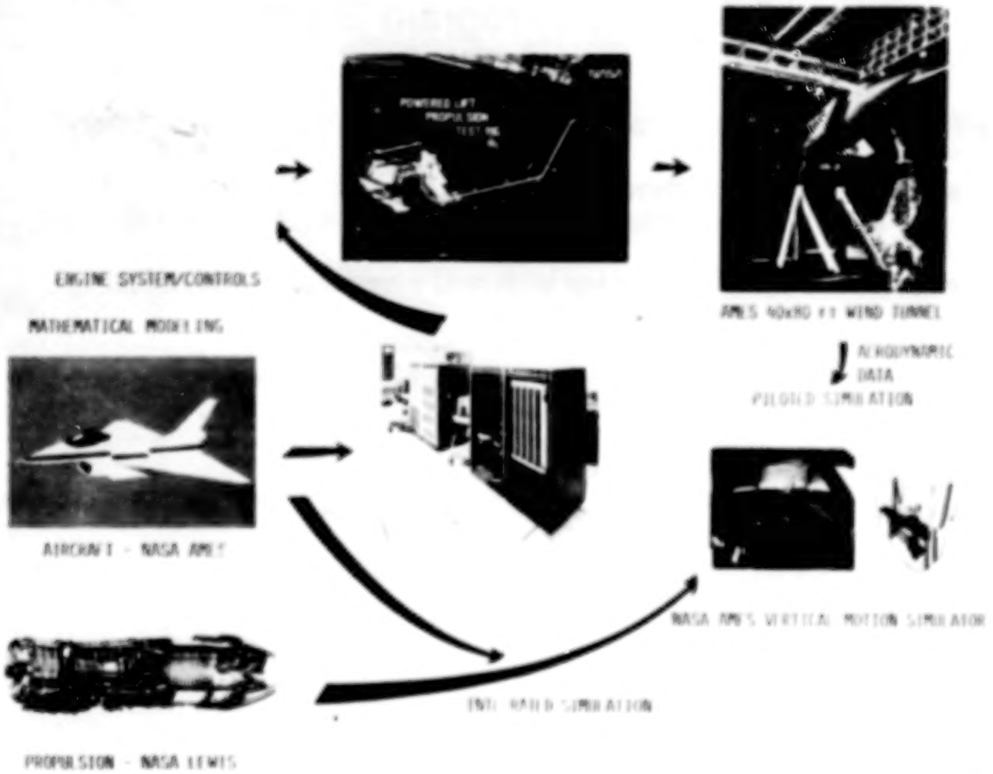
Integrated Methodology for Propulsion and Airframe Controls (IMPAC):

- Establish advanced integrated control methodologies for Supersonic STOVL aircraft
- Design candidate control concept and evaluate and validate system in piloted simulation and in ground-based experimental program

CD-91-54255

The Lewis Integrated Flight and Propulsion Control Program has two major components: (1) the E7D/F110 STOVL program, which is a contract effort with General Electric and General Dynamics and applies the Air Force-developed Design Methodology for Integrated Control (DMICS) and (2) an in-house effort called Integrated Methodology for Propulsion and Airframe Controls (IMPAC). Both efforts are coordinated with NASA Ames and plan validation tests on the Ames Vertical Motion Simulator.

INTEGRATED CONTROLS RESEARCH DEMONSTRATOR PROGRAM


POWERED LIFT
PROPULSION
TEST RIG
ENGINE SYSTEM/CONTROLS
MATHEMATICAL MODELING
AMES 40x80 FT WIND TUNNEL
AERODYNAMIC
DATA
PILOTED SIMULATION
AIRCRAFT - NASA AMES
NASA AMES VERTICAL MOTION SIMULATOR
INTEGRATED SIMULATION
PROPULSION - NASA LEWIS

ORIGINAL PAGE IS
OF POOR QUALITY

The NASA Lewis integrated controls research demonstrator program consists of two major paths for validating and demonstrating Integrated Flight Propulsion Control (IFPC) design methodologies for STOVL aircraft. One research path culminates in a piloted simulation of the target STOVL aircraft at the NASA Ames Vertical Motion Simulator. Piloted simulation will establish the handling qualities of a STOVL aircraft with a candidate IFPC concept. This work entails developing mathematical models for both the airframe and the propulsion system. The airframe model is formulated from aerodynamic data obtained from wind tunnel tests of the target airframe conducted by NASA Ames. The propulsion system model is based on existing gas generator systems augmented with STOVL-specific component data obtained by experimental testing conducted at NASA Lewis. The IFPC system was designed with control design methodologies developed both in-house at NASA Lewis and by industry.

The second research path involves propulsion system testing at NASA Lewis. This test involves actual propulsion system hardware, consisting of a current-technology gas generator and STOVL-specific, thrust-producing components (e.g., ventral nozzles and ejector nozzles). Computer-based, real-time models of the airframe and IFPC logic complete the overall test system. These tests will investigate the performance of a STOVL propulsion system with an IFPC. Propulsion system data obtained from this testing will be used to enhance the propulsion system models for the piloted simulation work. Together both the piloted simulation and the propulsion system testing will establish the validity of integrated control concepts for use in STOVL aircraft. (Funding for testing in the second research path has not yet been established.)

IMPAC—Integrated Methodology for Propulsion and Airframe Controls

Objective:

- **Develop and demonstrate through the in-house Lewis program a centralized IFPC design methodology**

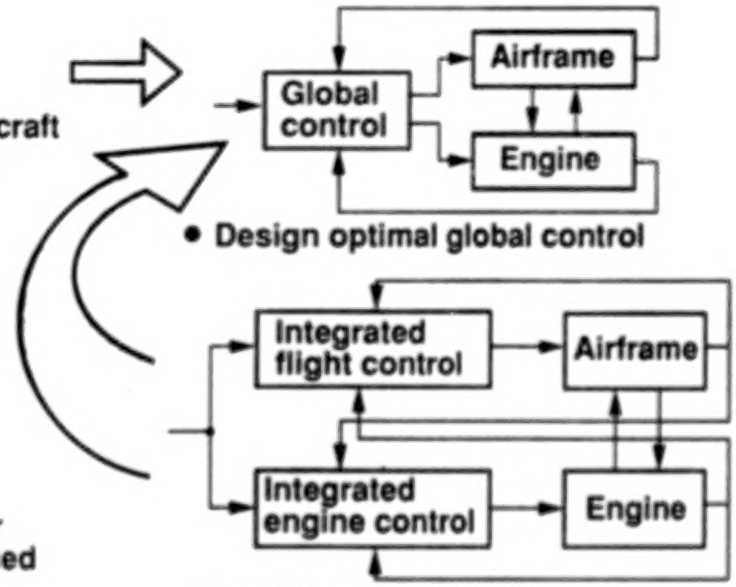
Method:

- **Demonstrate IMPAC design methodology for E7D SSTOVL aircraft through Lewis and Ames piloted simulation**

CD-91-54256

The objective of the IMPAC program is to develop and demonstrate a methodology for Integrated Flight and Propulsion Control (IFPC) design for future aerospace vehicles with a high degree of coupling between airframe and propulsion systems. The methodology to be developed will provide a viable alternative to the methodologies developed under the Air Force DMICS program while allowing for improved system performance and greater simplicity of control law synthesis and implementation. The three main phases of IMPAC are (1) methodology development, (2) IFPC design for the E7D SSTOVL aircraft, (3) methodology demonstration through piloted-simulation evaluation of the E7D aircraft on the Lewis and Ames simulators. Since both IMPAC and the Lewis-managed STOVL controls program are using the E7D test bed, the strengths and weaknesses of the IMPAC global and DMICS partitioned methodologies can be directly compared.

Integrated Methodology for Propulsion and Airframe Controls



- Develop integrated dynamic model
- Establish overall aircraft and engine control objective
- Design optimal global control
- Compare performance with global design
- Partition global controller
- Implement nonlinear aspects of partitioned controls

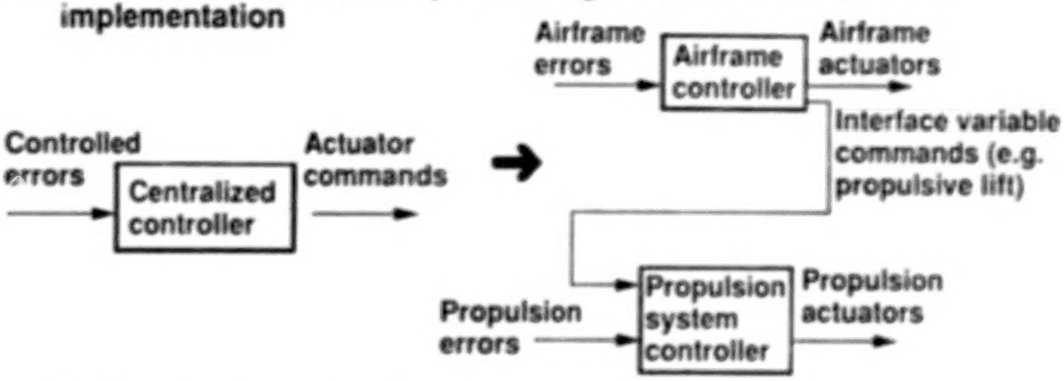
CD-91-54207

The IMPAC methodology consists of the following steps:

(1) Developing fully integrated dynamic airframe and propulsion system simulations, and generating linear models for control design
(2) Establishing the airframe and engine control objectives
(3) Performing the global control design to meet the control objectives
(4) Partitioning the centralized controller into separate airframe and propulsion system controllers for simplicity of implementation and independent subsystem checkout and performance accountability. This partitioning is to be done such that the integrated system performance with the partitioned controllers is close to the "optimal" performance with the centralized controller.
(5) Implementing the nonlinear aspects of the partitioned subcontrollers for operations over a wide range of flight conditions; incorporating safety limits and limit switching logic; etc.
(6) Evaluating the control design through closed-loop integrated nonlinear simulations and piloted simulations on fixed-base and motion simulators

Controller Partitioning

- Decentralized hierarchical partitioning for state-of-the-art IFPC implementation



Controlled errors

Centralized controller

Actuator commands

➔

Airframe errors

Airframe controller

Airframe actuators

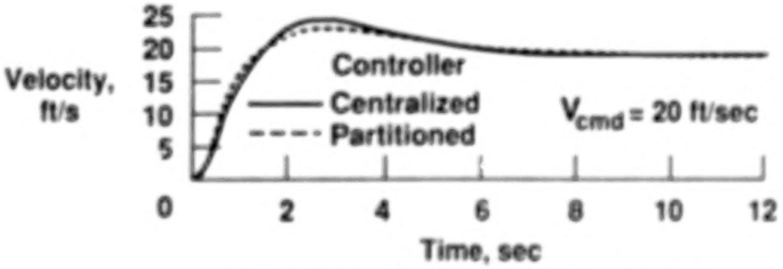
Interface variable commands (e.g. propulsive lift)

Propulsion errors

Propulsion system controller

Propulsion actuators

- Partitioning theory developed; computer algorithms under development
- Initial results encouraging; feasibility demonstrated



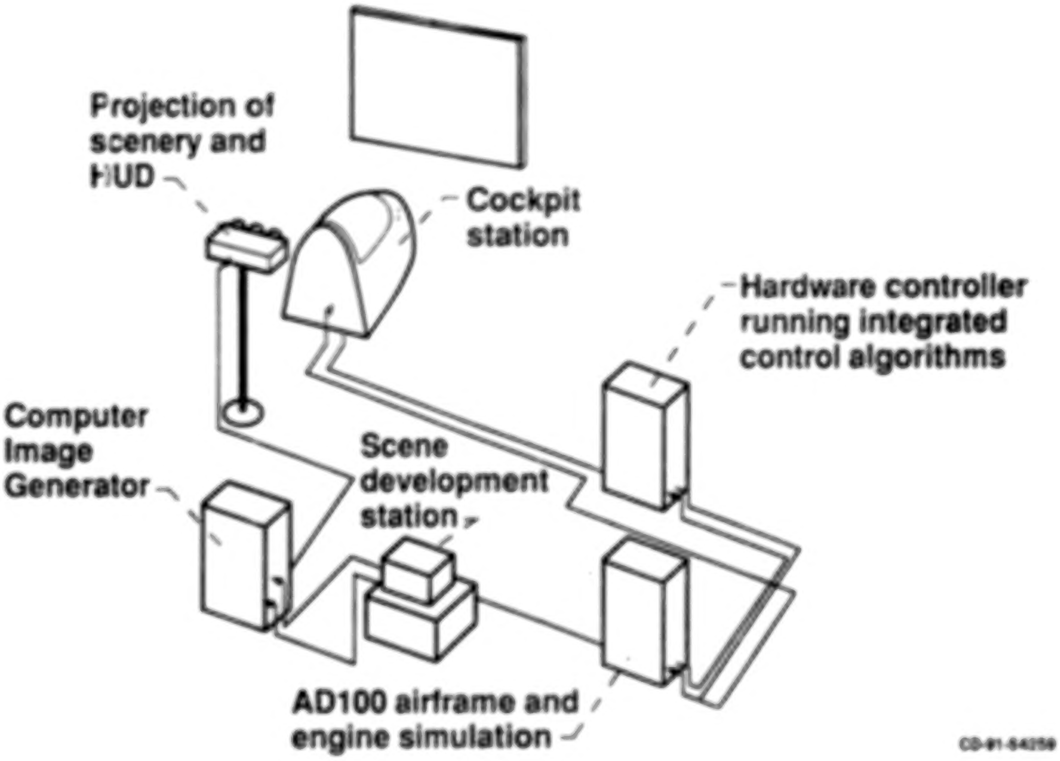
Velocity response for STOL aircraft

CD-91-54258

Although the centralized controller designed under IMPAC will be "optimal," it results in one high-order controller that is difficult to implement and crosses lines of corporate responsibility and expertise. Also the propulsion system manufacturer performs extensive tests with an independent subcontroller to ensure an adequate design. To address these difficulties, the idea of partitioning the centralized controller into separate airframe and propulsion subcontrollers has been introduced through the Lewis program. The partitioning "best" suited to IFPC implementation is one that results in a hierarchical structure for the propulsion subcontroller.

The theoretical solution of the decentralized, hierarchical controller partitioning problem for IMPAC is essentially complete. Numerical algorithms to implement the partitioning solution are currently being developed. Shown in the accompanying figure are responses of the IFPC system for a STOL aircraft to a step velocity command with a centralized controller and with an initial decentralized hierarchical controller obtained by partitioning. These initial results are quite encouraging and demonstrate the feasibility of the controller partitioning concept.

NASA Lewis Integrated Propulsion and Flight Control Simulator


Projection of scenery and HUD
Cockpit station
Hardware controller running integrated control algorithms
Computer Image Generator
Scene development station
AD100 airframe and engine simulation

CD-91-54259

The integrated propulsion and flight control simulator at NASA Lewis is a low-cost, real-time, fixed-base simulator used for integrated propulsion and flight control algorithm development, analysis, and evaluation. This simulator is used to evaluate both engine and airframe control performance in real time. With this evaluation capability the NASA Lewis flight simulation facility allows for rapid prototyping of integrated flight and propulsion control algorithms. Additionally, this simulator facility configuration provides easy, immediate access to engine and airframe data, interface variables, and control variables. Through the use of this flight simulator, integrated control algorithms are more readily tested, evaluated, and debugged prior to their evaluation on larger, motion-based flight simulation facilities.

The facility consists of a Computer-Generated Imagery (CGI) system; a single-channel projection system; a mockup fighter cockpit with sidestick controller, throttle, and rudder pedals; and a color touch screen monitor to emulate heads-down instrumentation in the cockpit. The AD100 simulation computer is used to drive the host engine and airframe simulations. The Control Interface Monitoring (CIM) unit executes the integrated control algorithms. The CIM unit is microprocessor based and fabricated in-house for the purpose of implementing and evaluating advanced digital control algorithms.

Status of Integrated Methodology for Propulsion and Airframe Controls

- Centralized design approach selected as primary thrust
- IMPAC methodology framework formulated
- Global controller partitioning theory and programs in development
- Effort to address nonlinear effects started
- Initial IFPC design for transition flight condition completed and published
- IMPAC methodology to be demonstrated in FY '91 in Lewis flight evaluation station

CD-91-54290

The IMPAC program has established the framework of a global integrated control design methodology. Partitioning methods have been established and computer programs are now being implemented to perform the required calculations. The goal now is to complete a control design including nonlinear effects for the E7D/F110 supersonic STOVL aircraft and to evaluate the design through real-time, pilot-in-the-loop simulation.

Life-Extending Control Concept

Control rates of change and levels of some performance variables to minimize damage of critical components while simultaneously maximizing dynamic performance

CD-91-54201

Systems with high performance requirements often have a small number of components that operate close to mechanical design margins. These critical components usually define the effective lifetime of the system and/or the mean time between maintenance actions. An active approach to managing the lifetime of these critical components can extend the time between maintenance actions or increase the total mission effectiveness of the system. This may be particularly true of systems that have long expected lifetimes, multiple mission objectives, and limited access to regular maintenance.

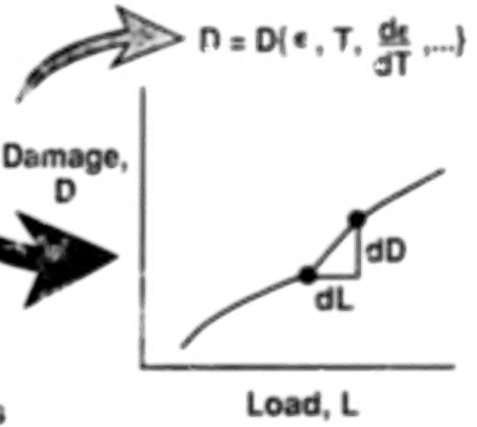
The concept of Life-Extending Control (LEC) means the management of the resource of component lifetime and the achievement of a desired objective. The desired objective is usually defined in terms of system performance. The concept depends on a prediction of the fatigue life of the particular component (or components) in the system. The component is assumed to be the life-limiting element or the most likely candidate for mechanical failure within the system. Currently, the fatigue life prediction of this component is based on a local, cyclic strain approach. The prediction of the remaining life of the component as well as an understanding of the effects of cyclic loading, stress, and strain on remaining life enable a quantification of the tradeoff between system performance and life to be established for a dynamic system. Once this tradeoff is defined, a control that maximizes system performance for a given lifetime of the component (or conversely, that minimizes damage for a given system performance) can be designed.

Continuous Life Prediction Approach

(Proposed)

Objective:

- Functionally relate differential damage with differential load (through local strain, stress, etc.)



$$D = D(\epsilon, T, \frac{d\epsilon}{dT}, ...)$$

Approaches:

- Derive from basic theory
- Empirically select expected forms and use optimization to best fit parameters

CD-91-54262

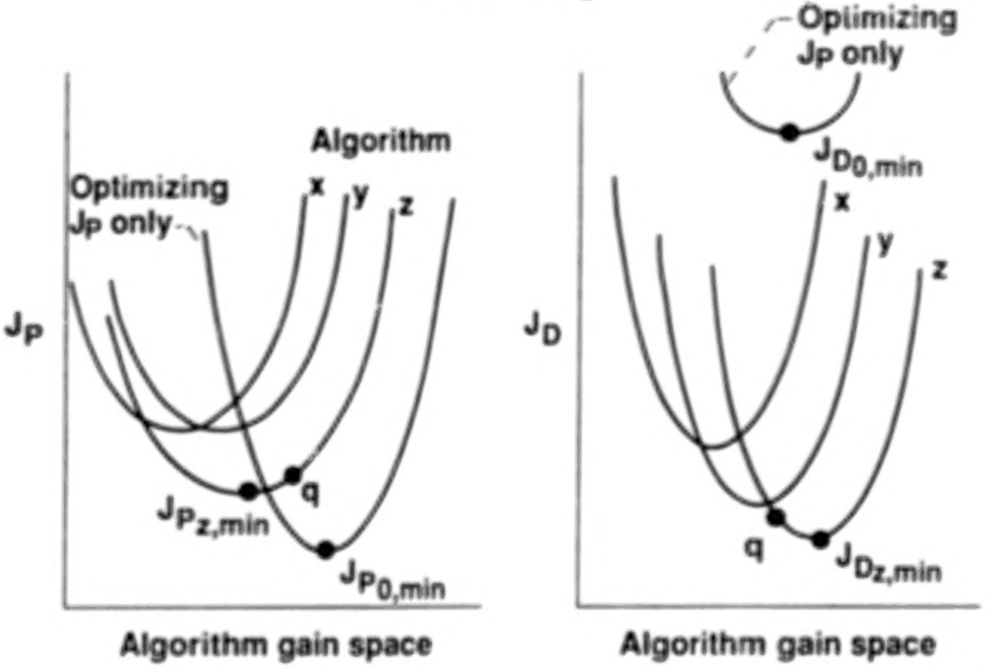
Current life prediction methods are cycle based. The cyclic life prediction process is highly nonlinear and comes from the study of metal fatigue due to cyclic loading. Usually the life estimates that result are used to forecast service life for a particular component design. The cyclic damage approach predicts damage over a stress-strain cycle based on material constants, average tensile stress, and cycle stress-strain amplitude. Total damage is then estimated by summing the cycle-by-cycle damage. Indirect life-extending control methods based on this type of damage model are now being studied.

However, the direct application of a life estimation procedure to control would be more beneficial and is therefore the ultimate goal of life-extending control. To accomplish this goal, a new formulation of the life estimation procedure is required. This new formulation will consist of a continuous form of the damage laws instead of the current forms, which require bookkeeping the number of cycles, their respective amplitudes, and their order of occurrence.

To achieve a continuous formulation of the life prediction process, an interdisciplinary approach is being used. Here the knowledge of material properties and life prediction of fracture and fatigue scientists must be combined with the control engineer's knowledge of dynamics and modeling to develop these continuous forms. The objective is to functionally relate measurable performance information with a differential form of the damage laws. This will allow the direct use of the differential estimate of damage in the life-extending control law and when integrated over complete cycles will give damage predictions that are equivalent (or perhaps superior) to those associated with the cyclic theory.

Effect of Various Life-Extending Control Algorithms on Performance J_P and Damage J_D

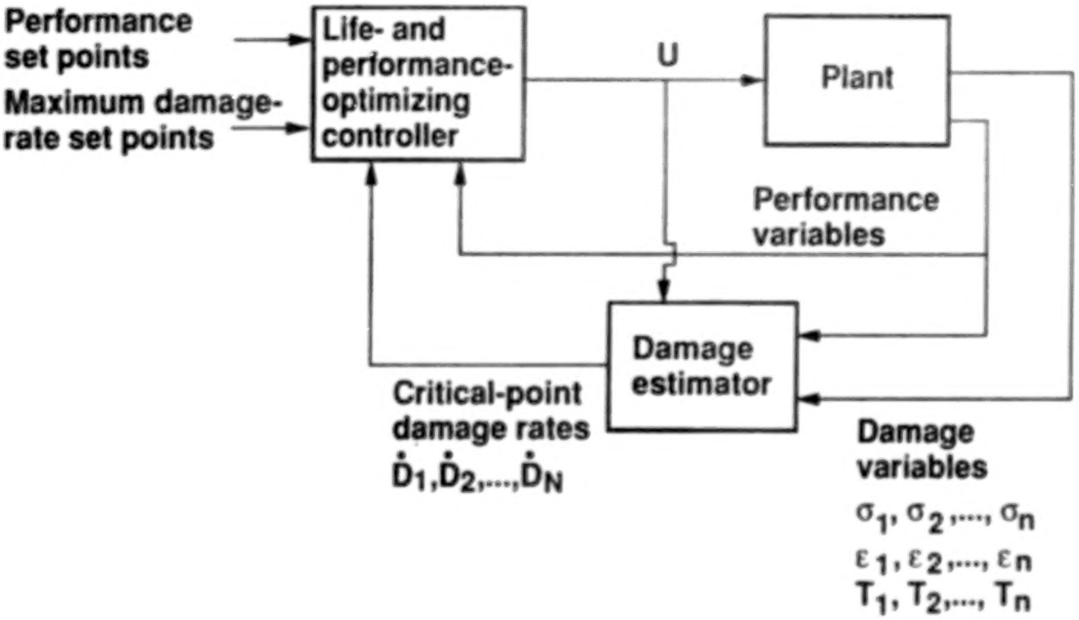
$$J = J_P + aJ_D$$


Optimizing
J_P only
Algorithm
x
y
z
J_P
$J_{P_{z,min}}$
q
$J_{P_{0,min}}$
Algorithm gain space
Optimizing
J_P only
$J_{D_{0,min}}$
x
y
z
J_D
q
$J_{D_{z,min}}$
Algorithm gain space

CD-91-54263

Two performance measures are considered: J_P, an objective function that maximizes dynamic performance, and J_D, a damage measure that uses the best (current) fatigue/fracture theory available to calculate the damage accumulated over the sequence of command transients. An overall performance measure can also be defined as $J = J_P + aJ_D$, where a represents the relative importance between performance and life extension. The basic concept is that designing a control for performance only would allow $J_{P_{0,min}}$ to be achieved. By introducing a good life-extending control algorithm, satisfactory dynamic performance $J_{P_{z,min}}$ (or q) can be achieved while significantly reducing damage from $J_{D_{0,min}}$ to $J_{D_{z,min}}$ (or q). Here, the point q is a compromise gain that allows good dynamic performance and damage reduction.

Life-Extending Control


Performance set points
Maximum damage-rate set points
Life- and performance-optimizing controller
U
Plant
Performance variables
Damage estimator
Critical-point damage rates
$\dot{D}_1, \dot{D}_2, ..., \dot{D}_N$
Damage variables
$\sigma_1, \sigma_2, ..., \sigma_n$
$\varepsilon_1, \varepsilon_2, ..., \varepsilon_n$
$T_1, T_2, ..., T_n$

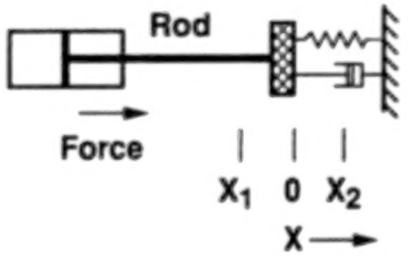
CD-91-54264

One of several Life-Extending Control (LEC) concepts that is currently being studied is shown in the figure. This concept assumes the availability of a continuous-damage model. Damage variables (stresses, strains, and temperature) are used to estimate the instantaneous rate of damage accumulation at selected critical points of the structure being controlled. The damage rates are then used by the control to optimize dynamic performance while minimizing damage rate.

Life-Extending Control

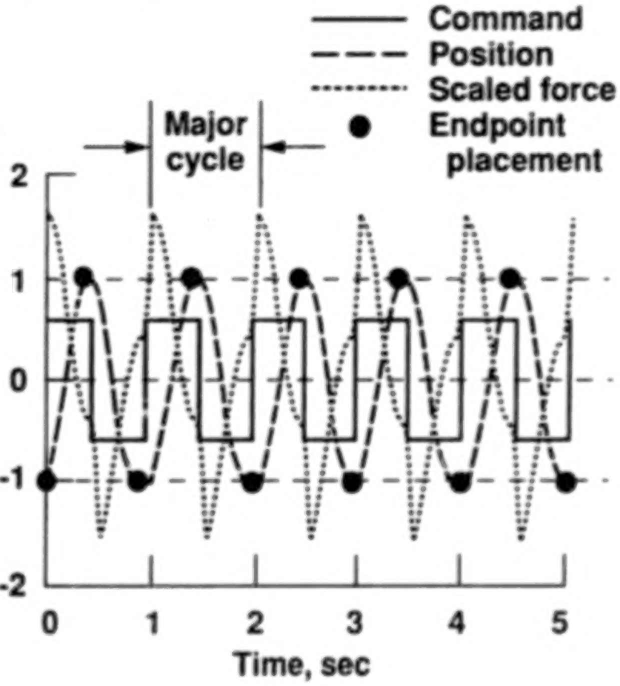
An Actuator Example

Hydraulic actuator position system



Damage (rod) = f (force) = f (N,T)

where
N number of times X = X_1 or X_2
T time to achieve N



Example results for case 1 trajectory command

CD-91-54265

A simple hydraulic actuator example for the positioning of a control surface is used to illustrate Life-Extending Control. The assumptions are that the actuator rod is the life-critical element and that its life is the resource to be managed while obtaining system performance. System performance is defined in terms of a positioning task. The mass at the end of the rod has to be placed, alternately, at positions X_1 and X_2. The number of placements (i.e., the number of times X equals X_1 or X_2) is N. Also, the time to achieve N is T. The performance is then defined to be

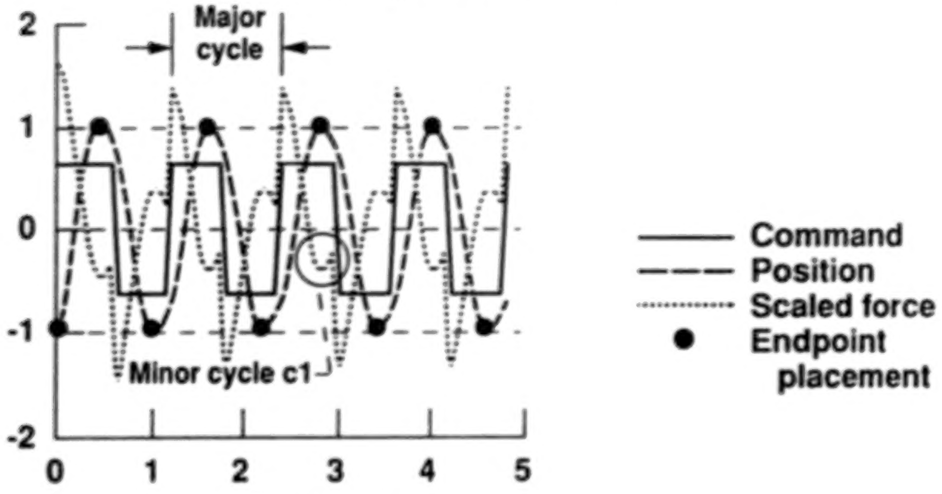
$$P = C_N N - C_T T$$

The first term in the equation is related to dynamic system performance, and the second term is related to accumulated damage. Here N is the number of times a command is given to move the control surface, T is the time required to achieve the desired commanded position, and the constants C_N and C_T represent the tradeoff between positioning accuracy and time to achieve the position. The life of the rod is a function of the compressive and tensile forces F applied to the rod.

Pulse sequence trajectory 1 was applied to the system. Also shown in the figure are the system position and scaled force trajectory resulting from pulse trajectory 1. The performance endpoints were selected as $X_1 = -X_2 = 1$. In this case $N = 11$ and $T = 4.9$ sec. A damage analysis was done on the simulated force trajectory. Here five major stress-strain cycles were observed and used to calculate damage. In this case $D = 0.0213$ units of damage, based on a total component life of 1 unit, was predicted.

Life-Extending Control

Example Concluded



Example results for case 2 trajectory command

Case	N	T, sec	N/T	F_{max}	Cycles	D	T_f	N_{total}	Performance
1	11	4.9	2.2448	1.6	5 Major	0.0213	230	516	802
2	9	4.6	1.9565	1.4	4 Major, 8 Minor	.0121	380	743	1106

Life-extending-control example results

CD-91-54266

A modified commanded pulse sequence trajectory, called trajectory 2, was applied to the same system. The commanded trajectory, the system position, and the scaled force for case 2 is shown in the figure. In case 2, $N = 9$ and $T = 4.6$ sec. The damage analysis in this case was based upon an observed count of four major stress-strain cycles and eight minor cycles. In case 2, $D = 0.0121$. Because the commanded pulse trajectory has been slowed slightly, the resultant force trajectory has smaller peak magnitudes. Consequently, the stress-strain cycles have smaller magnitudes and the damage is less.

Results obtained by assuming that the system will be operated to failure with the two pulse sequences selected are summarized in the table. The time to failure will be $T_f = 230$ sec for case 1 and $T_f = 380$ sec for case 2. The number of endpoint placements at T_f is $N_{total} = 516$ for case 1 and $N_{total} = 743$ for case 2. Given that the constants of performance are $C_N = 2$ and $C_T = 1$, the resultant performance is 802 for case 1 and 1106 for case 2. These performance constants can be interpreted as described above. Case 2 offers better total system performance by adopting a strategy that obtains endpoint placements at a slightly longer time, and results in a much more effective use of the lifetime of the critical component.

Additional Advanced Controls and Dynamics Activities

- Intelligent control of reusable rocket engines (neural-network-based control)
- Wave rotor dynamic modeling
- Reconfigurable control

CD-91-54267

Other activities are also being carried forward by the Advanced Controls Technology Branch. These are summarized here and include a very significant effort in Intelligent Controls for Reusable Rocket Engines. This effort is focused on the space shuttle main engine as a generic example.

Summary

Advanced aerospace controls have the potential to

- Optimize complex system performance
- Manage intersystem dynamic coupling to reduce pilot workload
- Simultaneously minimize damage to hardware and increase availability of hardware when damage does occur

The NASA Lewis Advanced Aeropropulsion Controls Program is structured to realize this potential.

CD-91-54268

The NASA Lewis Advanced Controls Program is focused primarily on propulsion system dynamics and controls. In a broad sense, this includes flight/propulsion control integration. Its objective is to bring advanced intelligence to complex aerospace systems, thereby achieving maximum steady and dynamic performance while improving system reliability and durability.

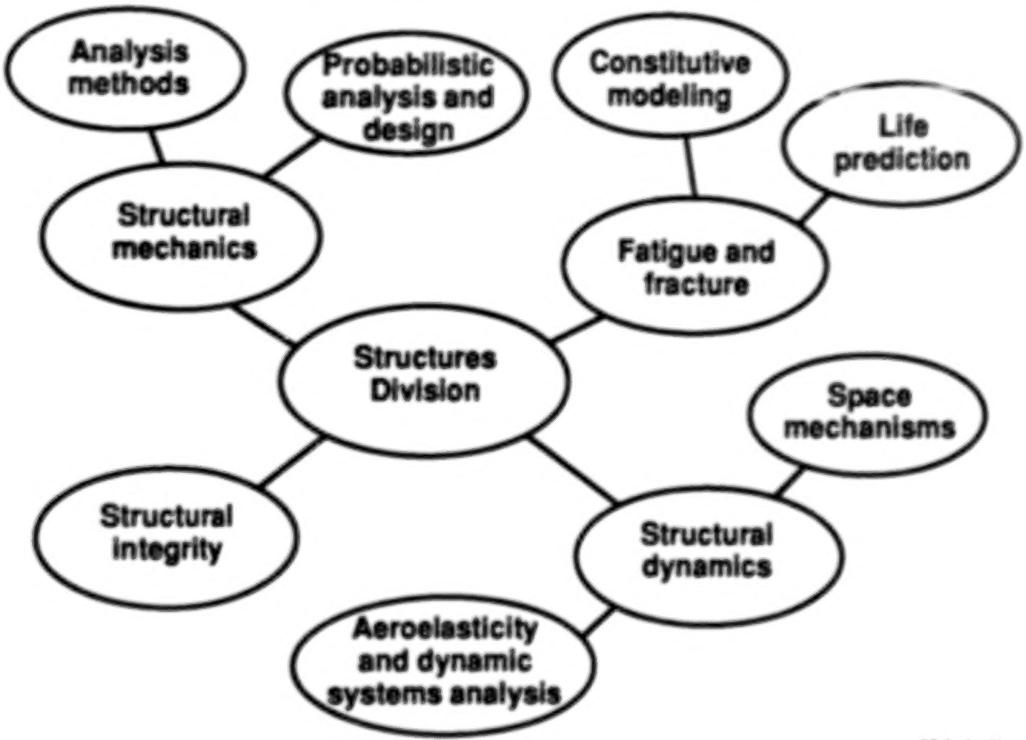
N91-20104

OVERVIEW OF STRUCTURES RESEARCH

Larry D. Pinson
NASA Lewis Research Center
Cleveland, Ohio

The development of aeronautical and space propulsion systems structures technology is the mission of the Structures Division. The technology required to achieve reliable, high-performance, lightweight structures needed for aerospace propulsion is among the most complex and challenging facing the design engineer. The division staff performs both fundamental and applied research in structural mechanics, fatigue and fracture, structural dynamics, and structural integrity. Research programs include probabilistic analysis and design, nonlinear material properties, symbolic logic, composite micromechanics, aeroelasticity, fatigue and fracture of composite structures, life prediction, and aspects of nondestructive evaluation. These programs, which for the most part are analytically based, are experimentally verified and are used to develop computer codes necessary for the design of complex engine structures. An overview of some of these programs is presented herein.

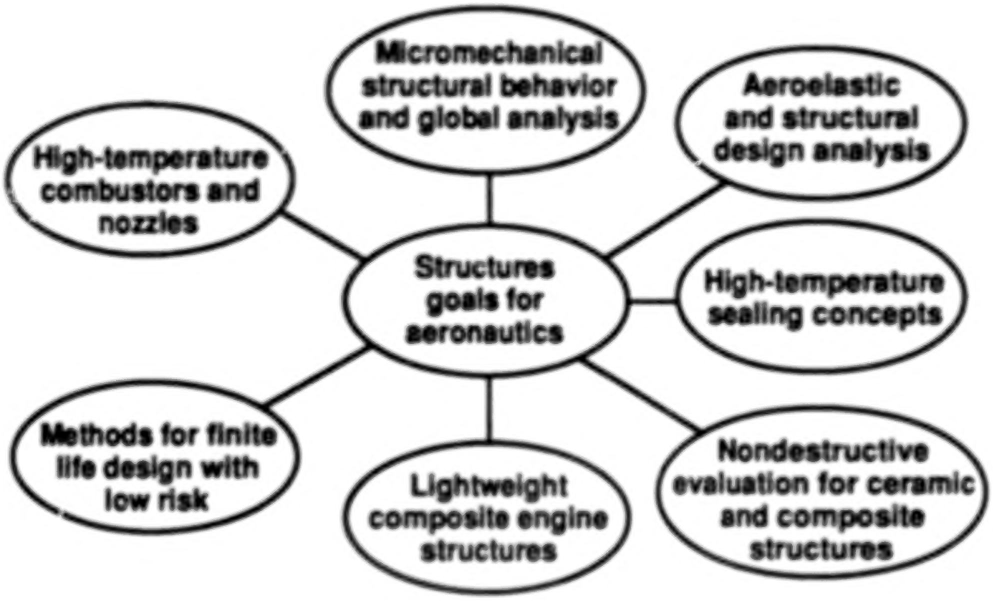
Structures Organization and Goals in Aeronautics


Analysis methods
Probabilistic analysis and design
Constitutive modeling
Life prediction
Structural mechanics
Fatigue and fracture
Structures Division
Space mechanisms
Structural integrity
Structural dynamics
Aeroelasticity and dynamic systems analysis

CD-91-[illegible]

The Structures Division supports and performs research in both space and aeronautical propulsion systems. With turbomachinery, large rotational kinetic energies can couple with vibration modes and result in large vibrations or dynamic instabilities. Prediction and control of these instabilities are major thrusts of our research. Metal matrix composites, ceramics, and ceramic matrix composites (CMC's) offer significant potential for higher thrust-to-weight ratios for gas turbine engines. This can be accomplished by allowing higher cyclic temperatures with the use of both metallic and refractory high specific strength material systems. These materials are especially of interest for 3000 °F combustors and nozzles. Emerging nondestructive

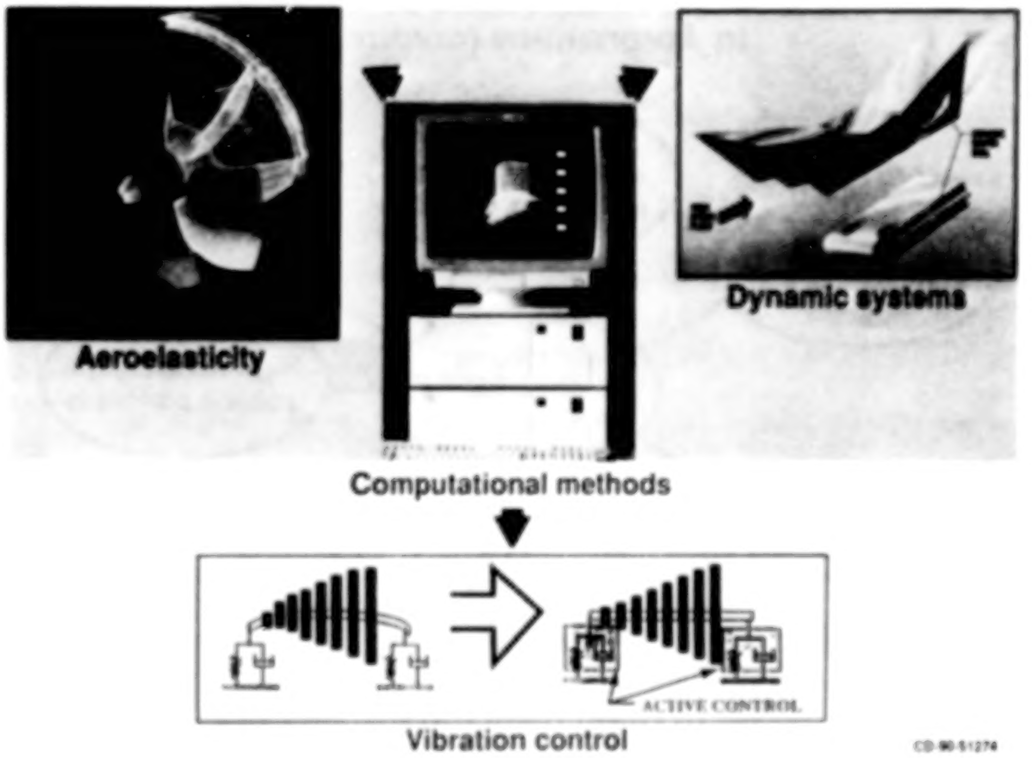
Structures Organization and Goals in Aeronautics (continued)


Structures goals for aeronautics
Micromechanical structural behavior and global analysis
Aeroelastic and structural design analysis
High-temperature sealing concepts
Nondestructive evaluation for ceramic and composite structures
Lightweight composite engine structures
Methods for finite life design with low risk
High-temperature combustors and nozzles

CD-91-[illegible]

evaluation (NDE) techniques may be used to verify the mechanical properties of these materials and to assess their degradation in service. Our approach is to develop what we term "analytical NDE" for characterizing materials factors that govern mechanical properties. We study micromechanical structural behavior. We have focused our activities on viscoplastic constitutive model development, experiments to calibrate and validate the models, and the development of nonlinear structural analysis methods and codes. The codes are used to predict the global behavior of the engine structure. Pioneering research is being performed in probabilistic methodology to predict structure performance, life, and reliability. This approach to analysis not only provides probable variations in the response but also the probability of occurrence. The probabilistic approach, in conjunction with finite-element analysis, couples laboratory coupon data with methods to design for finite life with low risk. The severe thermal environments under which hypersonic aircraft, such as the National Aerospace Plane (NASP), will operate require cooling and sealing of the engine walls, especially the combustor. Work is being undertaken in unconventional high-temperature sealing technologies.

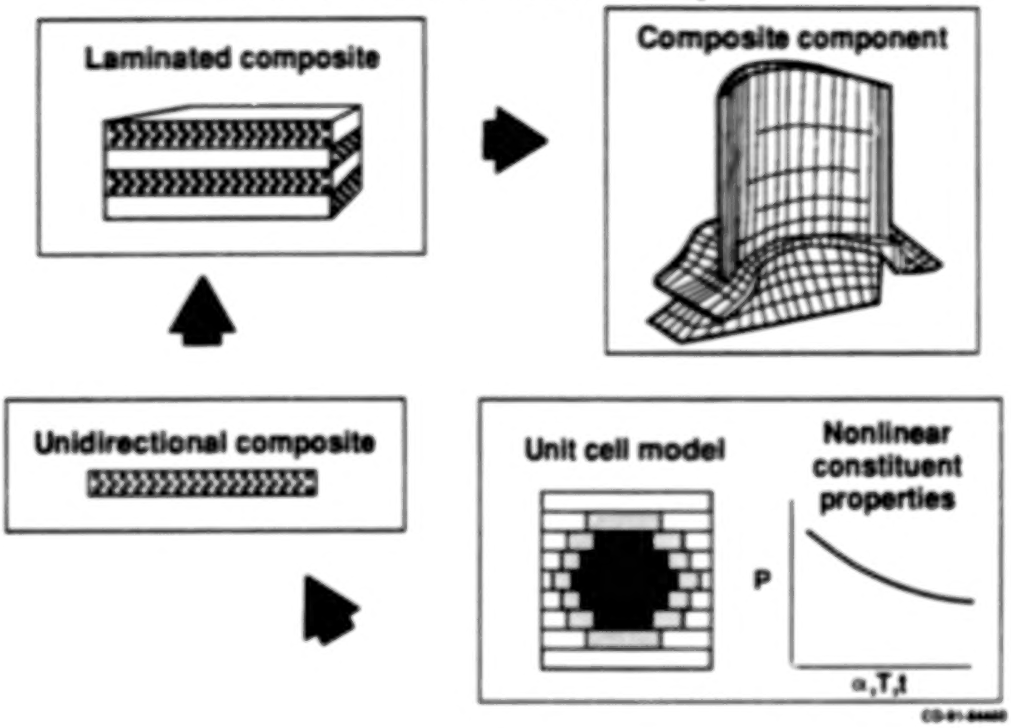
Aeropropulsion Structural Dynamics



CD-90-51274

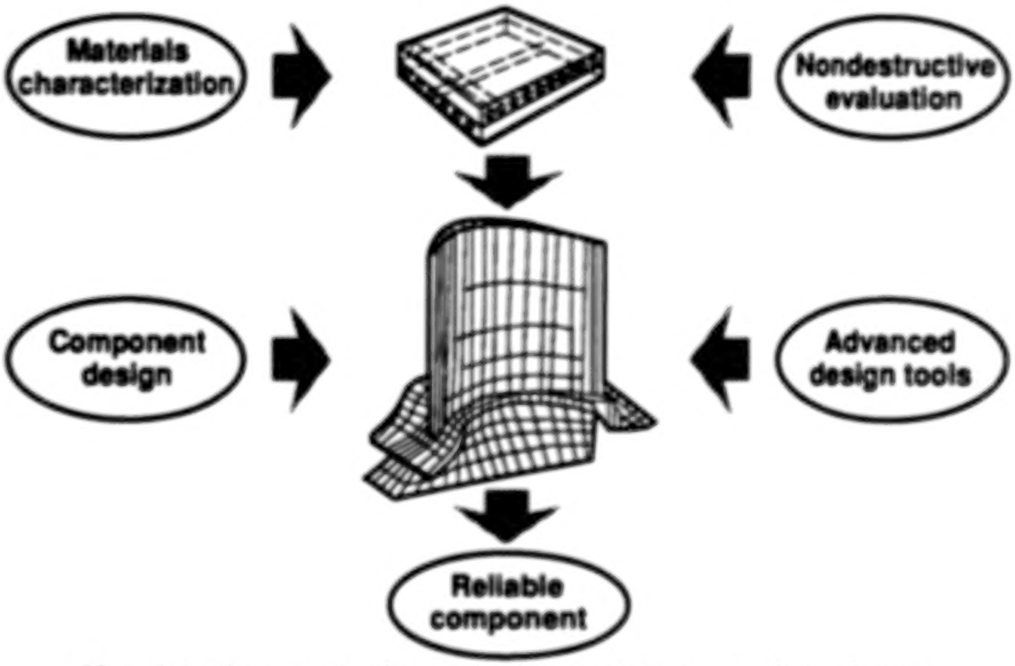
It is important to understand and define the physical interactions between thermal effects on basic material properties, the coupled aerostructural response (such as turbine blades made from such materials), and the coupling of the blade vibrations with net rotor-shaft rotor response. It is difficult, but not impossible, to simulate numerically these fundamental aeroelastic processes. In aeroelasticity, we are using both analytical and experimental means to define the performance limits of advanced propulsion systems, such as the ultra-high-bypass ducted fan engine shown here, and the supersonic fan, among others. Vibration control research is underway to develop and evaluate active control techniques and the required high-speed electronic controls to minimize unwanted shaft vibrations of both turbine engines and space turbopumps. Parallel computer processing is being used. The multigrid method, although used for years in fluid dynamics, now offers a new approach to nonlinear structural dynamics.

Composite Structures Simulation Relates Local Effects to Global Response



Mechanical performance and structural integrity of high-temperature metal matrix and ceramic materials is governed by local behavior at the level of the composite constituents, i.e., fiber and matrix. Hence, in the analysis and design of aircraft engine components which are ultimately to be fabricated from these composite materials, it is necessary to understand the local behavior of the material over the component volume and relate its effects to global structural performance. The critical local behavior is governed by such factors as imperfect bonding at the fiber-matrix interface, the progressive nature of microcracking, and nonlinear dependencies of constituent properties over the range of conditions in which the engine operates. We integrate constituent material models, cumulative damage models, composite mechanics, and global finite-element structural analysis (as depicted in the figure) to analyze this problem. With this capability, local effects on overall component behavior can be resolved and yet adequate efficiency achieved to be practical for realistic engine component applications.

Transition-to-Practice Technologies for Brittle Materials



Materials characterization

Nondestructive evaluation

Component design

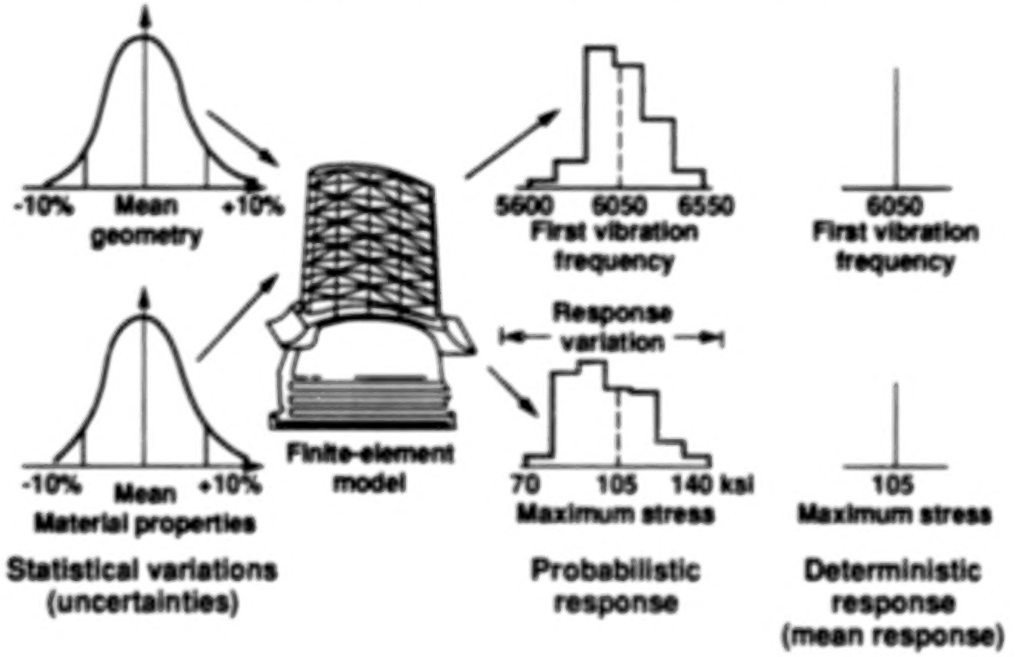
Advanced design tools

Reliable component

- **Materials characterization and advanced design tools are being used to develop ceramic components with high reliabilities.**

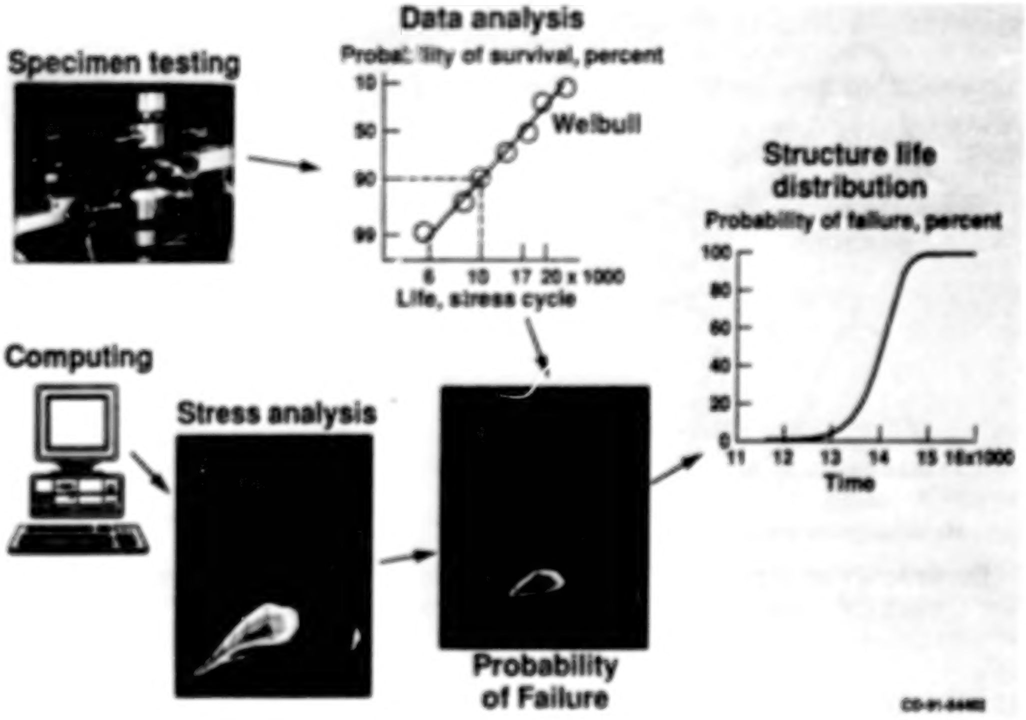
Transition-to-practice technologies are needed to bring brittle ceramic composite technology to the level necessary for practical application as aerospace materials. Two key areas where technologies are being used to develop ceramic composites are materials characterization and component design. Materials characterization techniques, both destructive and nondestructive, yield information on the material's failure mechanisms, integrity, and degradation. This crucial information guides and enhances the development of ceramic composites. Advanced design tools are also being developed to assist component designers with structural ceramic composite component development. These tools strongly influence the design of structural ceramic composite components by determining the component's reliability and life expectancy.

Probabilistic Analysis Methods Provide Decision-Making Basis for Structure Design


-10%
Mean
+10%
geometry
-10%
Mean
+10%
Material properties
Statistical variations
(uncertainties)
Finite-element
model
5600
6050
6550
First vibration
frequency
Response
variation
70
105
140 ksi
Maximum stress
Probabilistic
response
6050
First vibration
frequency
105
Maximum stress
Deterministic
response
(mean response)

In-house studies have demonstrated that probabilistic analysis methods under random loading are more reliable than deterministic approaches. Probabilistic methods have been used predominantly in fatigue, fracture mechanics, and structural reliability analyses under random vibrations. We have been developing probabilistic methods which synthesize statistical variations in loading, component geometry, material properties, and environmental conditions into a risk assessment for all levels of response. Contrarily, the deterministic approach provides either a mean or median value and does not cover any probable variations. As a result, even after using a safety factor, the risk may or may not be within acceptable limits. These analyses can be used for process quality control, inspection interval, maintenance schedule, qualification testing, and certification thrusts.

Life Prediction Methods Bridge Gap Between Specimen Testing and Full-Scale Engine Structure


Specimen testing
Data analysis
Probability of survival, percent
10
50
90
99
Weibull
6
10
17
20 x 1000
Life, stress cycle
Structure life distribution
Probability of failure, percent
100
80
60
40
20
0
11
12
13
14
15
16x1000
Time
Computing
Stress analysis
Probability of Failure

Using probabilistic methodology, the component design survivability can be mapped by incorporating finite-element analysis and probabilistic material properties. The method evaluates design parameters through direct comparisons of component survivability expressed in terms of Weibull parameters. The method allows the use of statistical data obtained from laboratory coupon testing under environmental conditions to be integrated into life and risk analysis of full-scale engine structures. It is possible through an interactive design process to minimize the risk of failure for a given operating time or, conversely, to design for a finite life for a defined risk. When Weibull parameters and the stress-life exponent of the material are unknown, it is permissible to assume these values in order to obtain a qualitative, if not quantitative, evaluation of a structural design. We are currently applying these methods to full-scale structures such as turbine blades and disks where full-scale component data exist.

CONCLUDING REMARKS

The development of aeronautical propulsion systems structures technology required for high-performance, advanced aircraft has been the mission of the Structures Division for many years. We carry out both fundamental and applied research in pursuit of this mission. We work cooperatively with industry and academia to apply the fundamental disciplines found in the university with the design and application needs of industry. It is from this perspective that the presentations were prepared.

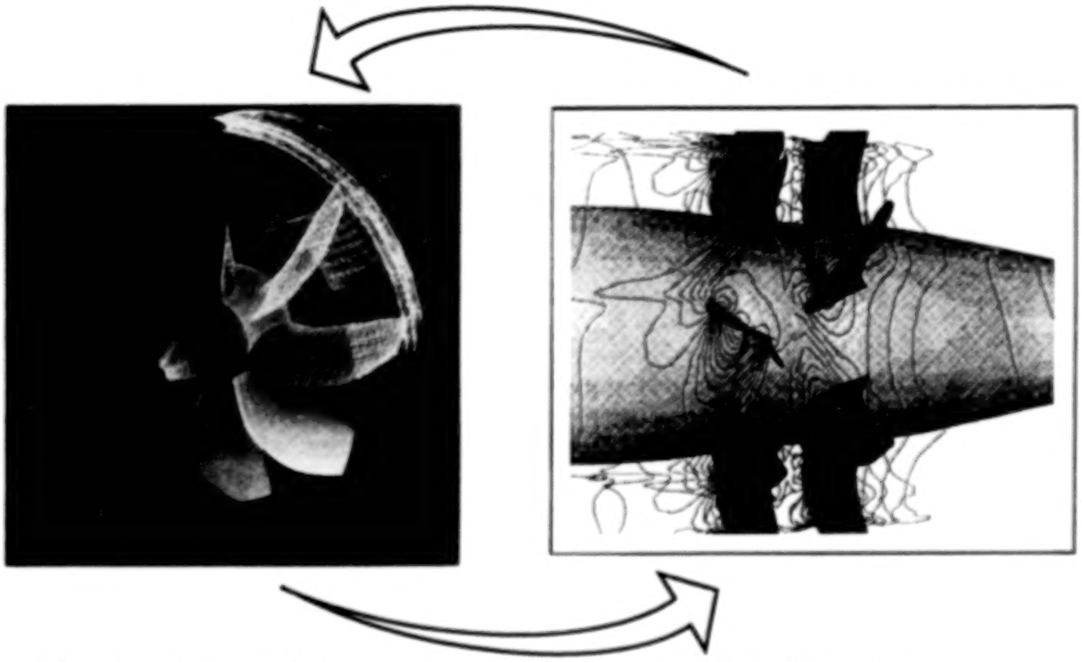
N91-20105

PROPULSION AEROELASTICITY, VIBRATION CONTROL,
AND DYNAMIC SYSTEM MODELING

Bruce M. Steinetz
NASA Lewis Research Center
Cleveland, Ohio

Aeropropulsion research in the Structural Dynamics Branch is aimed at addressing two key objectives: conceiving and implementing innovative structural concepts to enhance the performance of advanced aeropropulsion systems and developing and validating analytical techniques to define the limits of dynamic performance of advanced aeroengines, prior to costly full-scale testing. Aeroelasticity, vibration control, and dynamic systems, are the three research areas that make up the Structural Dynamics Branch. In the aeroelasticity technical area, researchers use both analytical and experimental means to extend and define the structural performance limits of advanced propulsion systems, including future ultra-high-bypass ducted turbofans, advanced turboprops, and high power-density turbopumps. Vibration control researchers are developing and evaluating active control systems and the required high-speed robust electronic controls to minimize unwanted shaft vibration of both turbine-engines and rocket engine turbopumps. Researchers in the dynamic systems technical area are developing a new class of high-temperature dynamic engine seals required for advanced hypersonic (e.g., NASP) engines, as well as developing advanced space mechanism technology to enable future space missions. Central to each of the branch's technical areas is the development of advanced computational methods which using modern computer science, will fundamentally improve solution speed and accuracy of large scale structural dynamics problems.

Aeroelastic Methods



Structural dynamics

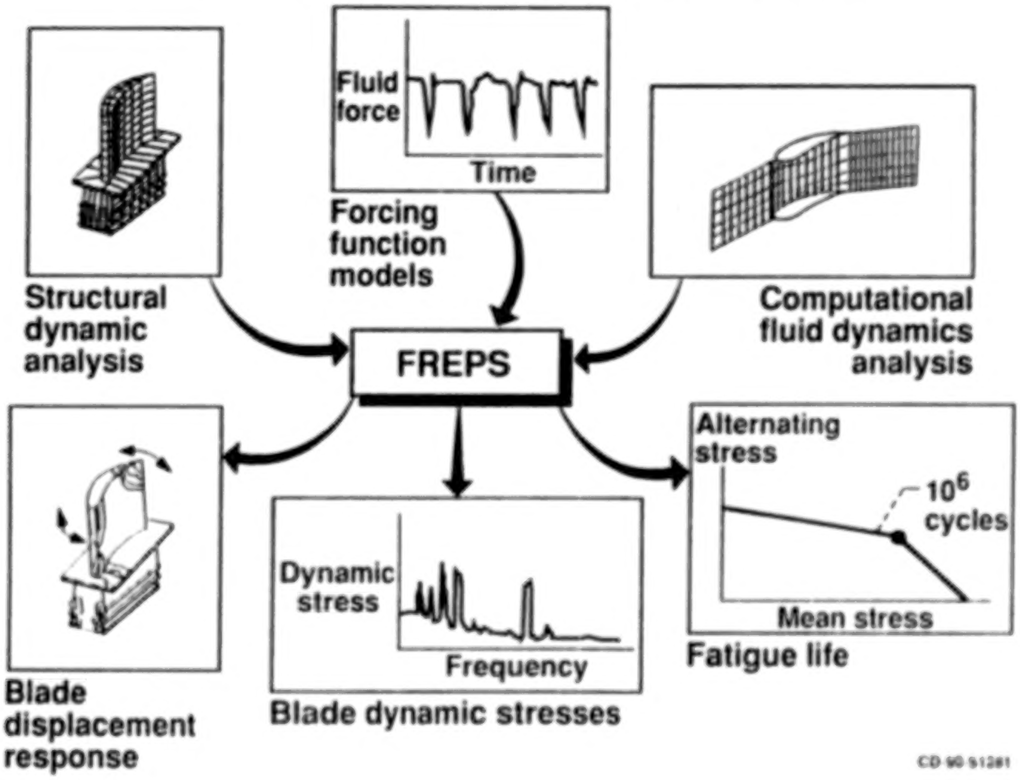
Aerodynamics

CD-90-51279

Aeroelastic methods are being developed and validated which can be used to define and in some cases extend the performance limits of aeropropulsion systems such as advanced ultra-high-bypass ratio ducted-fan engines and advanced turboprop engines. To predict the onset of the potentially dangerous condition of blade flutter (e.g., large amplitude blade vibrations caused by the interaction of unsteady air loads and blade natural frequencies), advanced computer codes which model the crucial blade/airstream interactions are being developed. Depending on the application, a wide spectrum of computer codes is used to define engine performance limits. The structural and aerodynamic models used in these codes increase in sophistication as the codes are developed. Examples of structural models used in increasing order of complexity are the spring-mass, continuous beam, and finite element representations. The geometry and materials used in modern blades require the finite element structural models for accurate analyses and are generally used for the final design.

The aerodynamic theories used model flow in the subsonic, transonic, and supersonic flow regimes. Examples of steady and unsteady cascade aerodynamic codes being developed in increasing order of complexity include the panel, full-potential, Euler, and Navier Stokes methods. The aeroelastic analyses are being developed to account for the effects of mistuning (e.g., small variations of blade-to-blade natural frequencies, blade angle of attack, etc.) that can lead to dangerous operating conditions in next-generation, high-performance blades.

Turbomachinery Blade Forced REsponse Prediction System

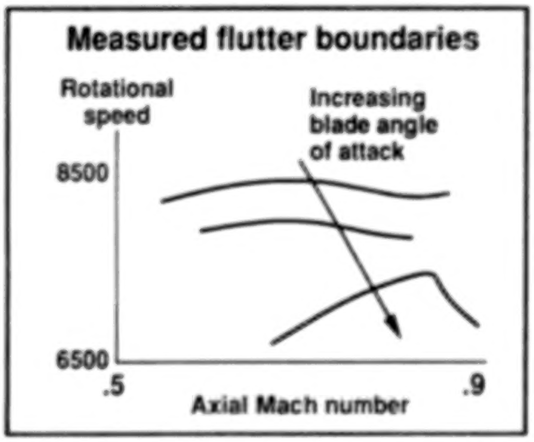

Fluid force
Time
Forcing function models
Structural dynamic analysis
FREPS
Computational fluid dynamics analysis
Alternating stress
10^6 cycles
Mean stress
Fatigue life
Dynamic stress
Frequency
Blade dynamic stresses
Blade displacement response
CD-90-51281

Once the dynamic blade performance limits have been defined, structural designers must then determine the dynamically induced blade stresses to enable improved prediction of blade fatigue lives.

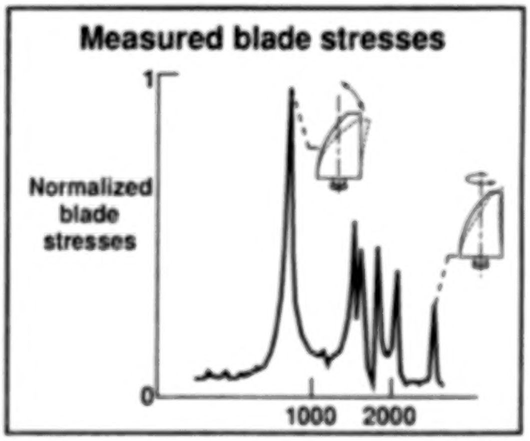
A methodology to conduct this forced response analysis of highly loaded and cambered turbomachinery blades called FREPS (e.g., Forced Response Prediction System) is being developed by the Structural Dynamics Branch. This modal-based finite element method combines two-dimensional steady potential-flow and linearized unsteady potential flow with solid finite element structural analysis. After the blade natural frequencies and mode shapes have been computed, FREPS then calculates blade displacement response and cyclic blade stresses. Knowing the cyclic and steady blade stresses, designers can estimate through the use of the Goodman diagram the anticipated blade life. To validate the key elements of this methodology, experiments are planned to measure unsteady aerodynamic forcing functions and blade response.

FREPS is currently being applied to analyze the highly loaded, life-limited space shuttle main engine high-pressure-oxygen-turbopump blades that carry on the average 300-hp per blade.

Experimental Aeroelastic Research



Measured flutter boundaries



Measured blade stresses

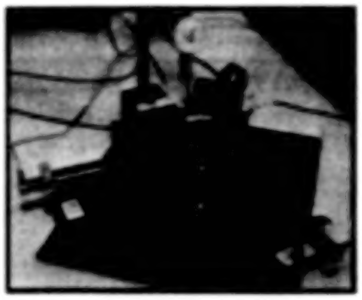
CD-90-51282

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

In conjunction with the engine industry and the Propulsion Systems and the Aeropropulsion Facilities and Experiments Divisions here at Lewis, experimental aeroelastic research is conducted to gain first-hand understanding of blade aeroelastic phenomena. The physical insight gained by the wind tunnel tests guides the analytical development, and the data collected are used to validate the complex analysis procedures. Blade performance parameters such as flutter boundaries and blade stresses are measured while operating scale propulsor models under simulated flight conditions in Lewis' wind tunnels and in a specialized vacuum spin rig. For instance, using advanced instrumentation techniques, such as laser-based blade vibration measurement system, flutter boundaries were measured for a counter-rotating propfan, as shown on the left.

Experimental research in the Structural Dynamics Branch is also addressing aeroelastic issues associated with energy efficient counterrotation propfans being considered for next generation cruise missiles. As shown in the figure on the right, blade vibrational mode shapes, natural frequencies and stress levels are determined through advanced experimental methods such as laser holography.

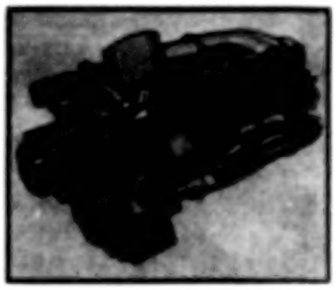
Active Control Of Rotor-Shaft Dynamics



Piezoelectric bearing actuators

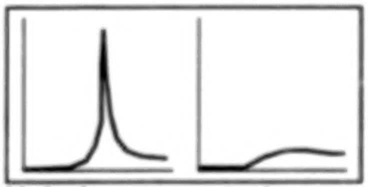
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Transient rotor dynamics rig

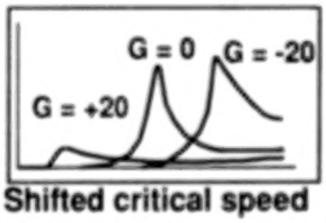


Engine demo

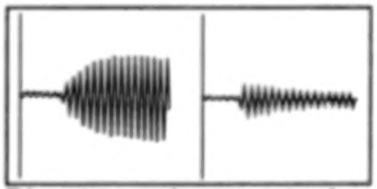
Test data



Unbalance control



Shifted critical speed



Direct transient control

CD-90-51283

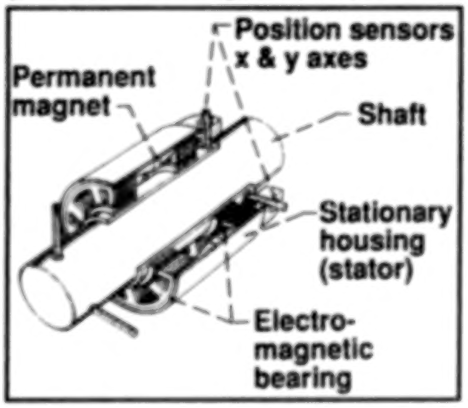
Two types of active control techniques are being pursued in the Structural Dynamics Branch to minimize unwanted shaft vibrations caused by unbalance, transient effects, or supercritical shaft-speed operation. In the first technique, piezoelectric actuators exert modulated control forces on the turbine shaft through soft-mounted shaft bearings.

In laboratory tests the piezoelectric actuators have been shown to be very effective in controlling the vibration of high speed shafts. For instance the peak vibration of a turbine shaft when passing through the first bending critical was reduced by a factor of 4. In another test, tuning the actuator/controller system to effect a different bearing stiffness allowed the shaft's critical speed to be shifted either above or below the normal critical speed by up to 50 percent. This feature of active control permits engine design strategies where operation of the shaft at critical speeds is not necessary. Using the transient rotor-dynamics rig shown in the figure, vibration response to transient events such as might occur during a blade loss or hard landing were reduced five-fold.

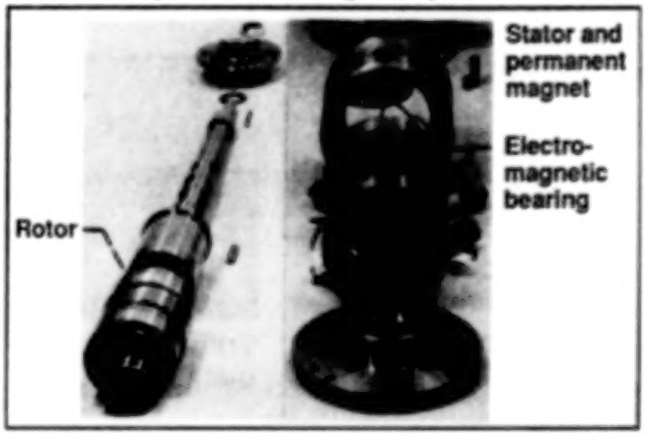
The above benefits of active vibration control can translate into substantial performance gains and weight savings in future turbomachines, removing vibrational constraints that presently limit engine design. Through a cooperative program with the Army, Allison, and Texas A&M, a T-63 helicopter demonstration engine is to be retrofitted with the piezoelectric shaft control system to quantify the installed dynamic performance benefits.

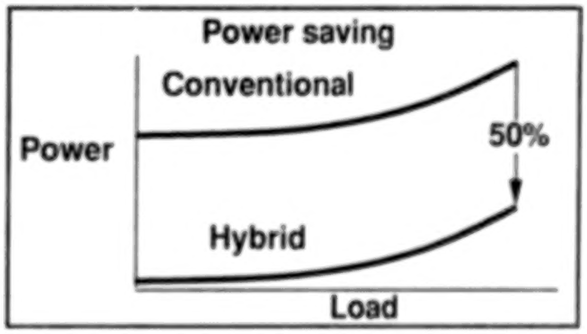
Hybrid Magnetic Bearing for Cryogenic To 400 °F Applications

Magnetic bearing with permanent magnet (PM) bias



Magnetic bearing components




Power saving

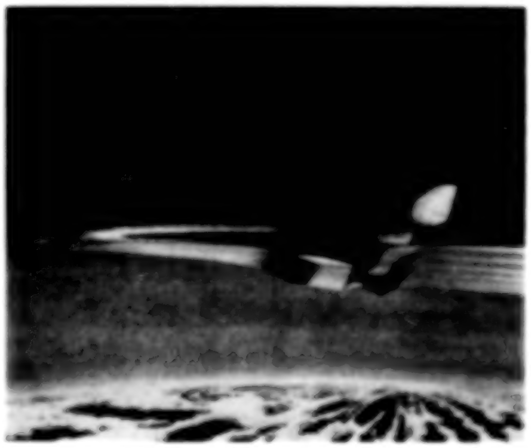
CD-90-51284

The second class of active vibration control techniques under development at Lewis is the magnetic bearing. Previous impediments in applying magnetic bearings to aerospace applications were the size, complexity, and weight of their control and power systems and the physical size of the magnetic bearing. Developments in control and power electronics have removed these barriers.

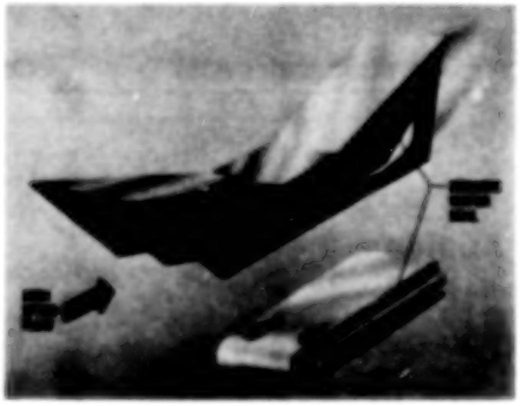
Magnetic bearings in turbomachines offer wear elimination, performance improvements, and potential weight savings. These advantages are achieved since there is no contact between the rotor and the bearing and since no lube system is required. Also, the stiffness and damping of the bearing can be varied, and the limits on shaft speeds and diameter can be substantially extended. Shaft vibrations can be minimized by properly modulating the electromagnetic forces applied to shaft through advanced digital controls, also under development.

Current research underway at Lewis is aimed at improving bearing load capacity and reducing bearing power consumption through implementation of hybrid bearing concepts such as the permanent-magnet-biased, magnetic bearing shown in the figure. In this concept the permanent magnets work in conjunction with the electromagnets to suspend the shaft. Relative to the conventional magnetic bearing (e.g., without the permanent magnet bias) the hybrid bearing reduces by 50 percent the power required to suspend and control the shaft under load. This concept produces less rotor heating which is important for very high-speed shafts. The other key advantage of this hybrid bearing design is the high radial load capacity. The design radial load capacity of the bearing shown is 500 lb for a 1-in.-diameter shaft. Depending on the final application, the magnetic bearing shown can operate over a broad temperature range of cryogenic to 400 °F.

National Aerospace Plane Engine Seals



Single stage-to-orbit
Mach 25 vehicle



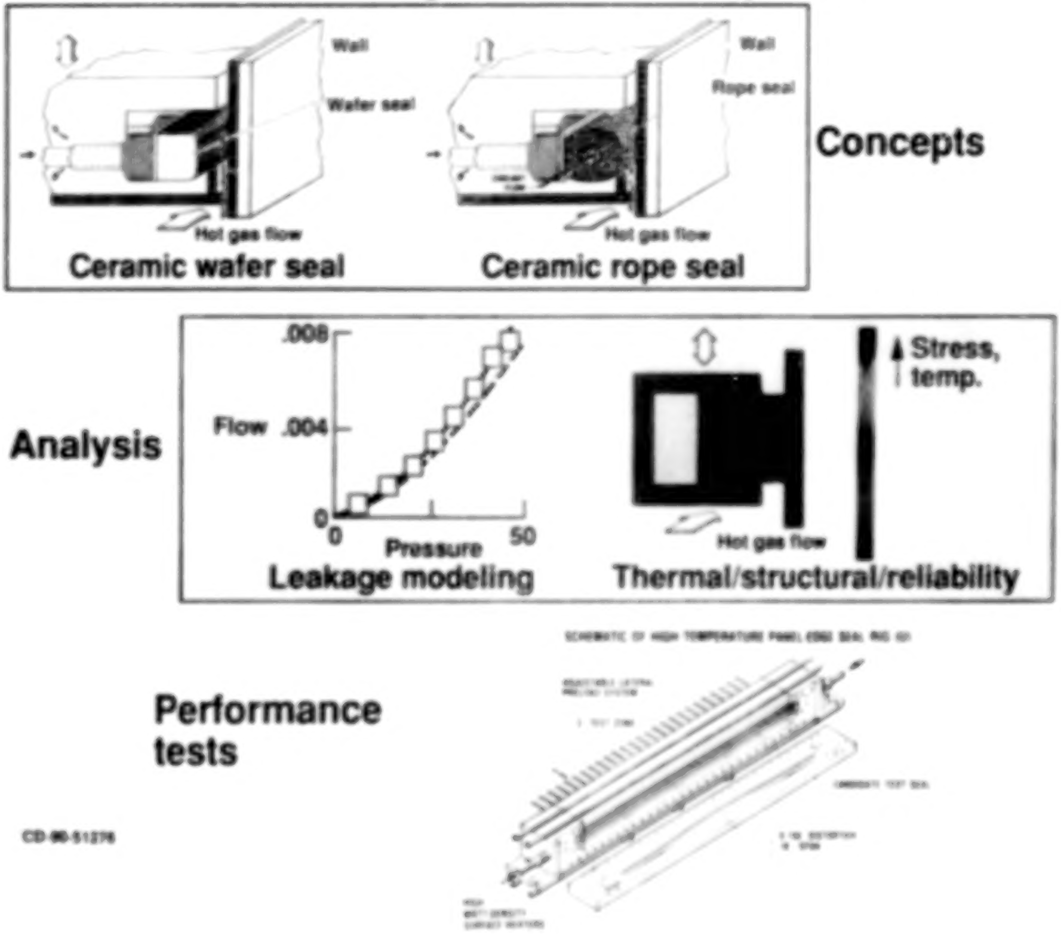
High temperature
dynamic engine seals

CD-90-51275

Key to the successful development of the single-stage-to-orbit National Aerospace Plane (NASP) is the successful development of combined cycle ramjet/scramjet engines that can propel the vehicle to 17 000 mph to reach low Earth orbit. To achieve engine performance over this speed range, movable engine panels similar to those used in advanced fighter jet two-dimensional exhaust nozzles are used to tailor engine flow. NASA Lewis is developing a family of new high-temperature seals to form effective barriers against leakage of extremely hot (>2500 °F), high pressure (up to 100 psi) flow path gases containing hydrogen and oxygen. Preventing backside leakage of these explosive gas mixtures is paramount in preventing the potential loss of the engines or the entire vehicle.

As shown in the engine isometric, the primary function of this sliding seal is to seal the many feet of linear gaps between the articulating horizontal engine panels and the adjacent vertical engine walls. Complicating the seal's mission is the need to accommodate and seal significant adjacent wall distortions (up to 3/16 in. in only 18 in. of seal length) caused by pressure and thermal loads on these weight minimized panels.

NASP Engine Seal Development


Wall
Wafer seal
Hot gas flow
Ceramic wafer seal
Wall
Rope seal
Hot gas flow
Ceramic rope seal
Concepts
Analysis
.008
Flow .004
0
0
Pressure
50
Leakage modeling
Stress, temp.
Hot gas flow
Thermal/structural/reliability
Performance tests
CD-90-51276

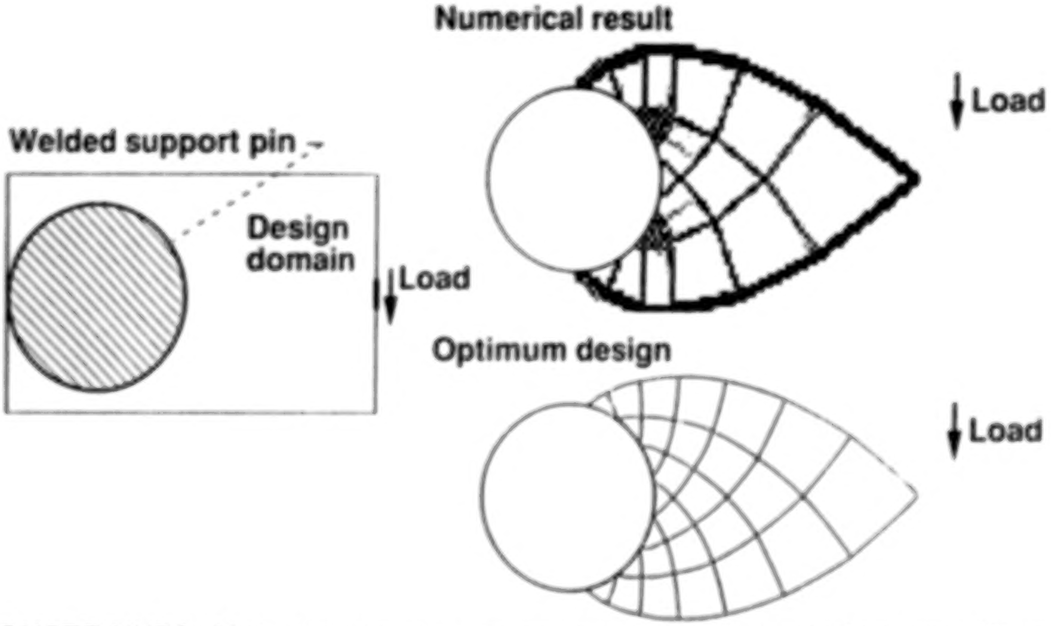
Advancements in state-of-the-art high temperature engine seals are being made by (1) developing advanced seal concepts fabricated from engineered ceramic materials, (2) developing and applying advanced analytical techniques to assess seal performance over the broad spectrum of engine operating conditions, and (3) finally demonstrating seal performance using specially designed high temperature test fixtures.

Two leading NASP engine seal candidates are the ceramic wafer seal and the braided ceramic rope seal shown. Both seals can operate hot to temperatures up to 2000 °F without coolant. The ceramic wafer seal derives its flexibility through relative sliding of the adjacent seal wafers. Low leakage, braided ceramic rope seals are made using advanced braiding techniques, under development, that take advantage of the high temperature flexibility of Nextel ceramic fibers. Analytical techniques recently developed under this program show promise in allowing seal designers to predict leakage flow through these braided preform seals as a function of engine operating conditions . Also, Lewis has developed and applied advanced iterative finite element techniques to assess wafer-seal thermal/structural and reliability performance under the extreme heating rates (up to 1160 Btu/ft^2-sec) expected during hypersonic flight.

Performance tests conducted at NASA Lewis have successfully demonstrated seal operation at engine simulated temperatures (up to 1350 °F), and pressures (up to 100 psi) sealing both flat and expected wavy wall conditions

Structural Shape Optimization

PROBLEM: Find minimum weight structure.
TEST CASE: Classic cantilever beam with overhung load.



CONSTRAINTS: Material strength, component weight, boundary conditions.

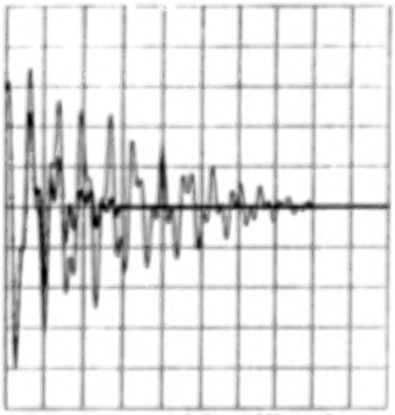
CD-90-51277

As part of the computational methods effort within the branch, automated techniques to perform structural optimization are under investigation in conjunction with the University of Michigan. A major difficulty of shape optimization of structures is caused by large domain changes during the optimization process that requires the rediscretization of the domain to conduct the finite element stress analysis. Furthermore, changes of topology, such as introduction of weight-saving holes in the structure, are virtually impossible with existing optimization approaches, unless done manually. To resolve these difficulties and limitations of traditional shape optimization, a new method has been developed. Optimization and discretization are performed in a fixed domain using a unit-cell approach to automatically generate the optimal shape and topology from a fixed domain, subject to given load and support conditions. The design domain is discretized into a grid-work pattern of unit-cells whose areas are modified sequentially until the compliance of the structure is minimized subject to a user-defined weight and material strength constraints. The unit-cell areas are tailored during the optimization using a percent-void parameter that varies between 0.0 for a heavily loaded (e.g., full) unit-cell and 1.0 for a lightly loaded (e.g., empty) cell.

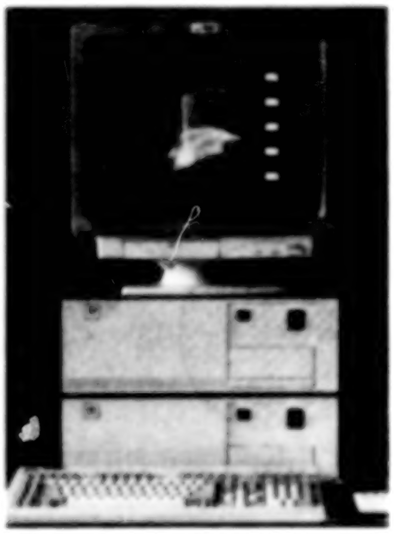
The method was successfully demonstrated using the classical test case of a cantilever beam with an overhung load. The Michell truss shown in the lower right corner of the figure is the optimal (e.g., minimum weight) truss structure for carrying bending loads to a circular attachment pin. The structural shape predicted by the unit-cell shape optimization technique is shown in the upper right corner of the figure and bears a remarkable resemblance to the optimum. Structural designers using optimal shapes predicted from this method can easily mentally smooth the slightly irregular pattern and quickly complete the component's final design.

Dynamic Systems Analysis

Structural substructuring



Parameter identification



Hardware advancements (e.g., transputer)

CD-90-51278

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

To improve the speed and performance of structural dynamic computations, the researchers in the dynamic systems group are making advancements in the areas of (1) structural-dynamic substructuring methods, (2) parameter identification methods, and (3) computational methods. In the area of substructuring, specially developed boundary flexibility methods of component mode synthesis permit accurate dynamic calculations with significant reductions in computational time. Using parameter identification techniques which allow experimentally determined properties (e.g., stiffness and damping) to be entered into the numerical model allows much more accurate dynamic system performance calculations.

In the area of computational methods, advanced hardware architectures (such as parallel processors, neural networks, and transputers) along with computational strategies are being used to greatly improve solution efficiency of structural dynamic analysis problems.

For instance, Branch researchers using a specially developed 32-processor transputer demonstrated analysis times one-third that of a Cray X-MP24 in performing a finite element analysis of an SSME turbine blade. This speedup, combined with the relatively low-cost of the transputer system, gives this table-top workstation a performance-cost ratio of about 60 times better than the Cray X-MP24 system.

SUMMARY

Accomplishments and research-in-progress in the area of aeropropulsion have been reviewed for the Structural Dynamics Branch. Researchers in the areas of aeroelasiticty, vibrations control and dynamic systems are conceiving and implementing innovative structural concepts to enhance performance of advanced aeropropulsion systems and are developing and validating analytical techniques to define the limits of dynamic performance of advanced aero-engines, prior to costly full-scale testing.

Sophisticated aeroelastic methods under development are used to perform interactive fluid and structural analyses to define the performance limits (e.g., such as onset of blade flutter) of advanced turbine engines under both steady and unsteady conditions. Newly developed blade-forced response analyses go beyond defining performance limits and are being applied to predict blade dynamic displacements and blade cyclic stresses, allowing prediction of blade fatigue life.

To control unwanted shaft vibrations of advanced turbine engines vibration control researchers are developing two active-control techniques: piezoelectric actuators and magnetic bearing actuators. Combining high speed digital controls and advanced vibration sensing techniques with either of these active-control devices enables extension of engine performance limits and offers the new capability to squelch shaft vibrations caused by otherwise damage-producing transient conditions (e.g. hard landings or blade loss).

Researchers in the dynamic systems group are developing a new class of high temperature seals that show promise of meeting the demands of advanced hypersonic engines (e.g., NASP). Researchers are also developing and implementing advanced algorithms and computing hardware to improve solution speed, accuracy, and costs of large structural dynamics problems.

N91-20106

COMPUTATIONAL SIMULATION OF PROPULSION STRUCTURES PERFORMANCE AND RELIABILITY

Dale A. Hopkins
NASA Lewis Research Center
Cleveland, Ohio

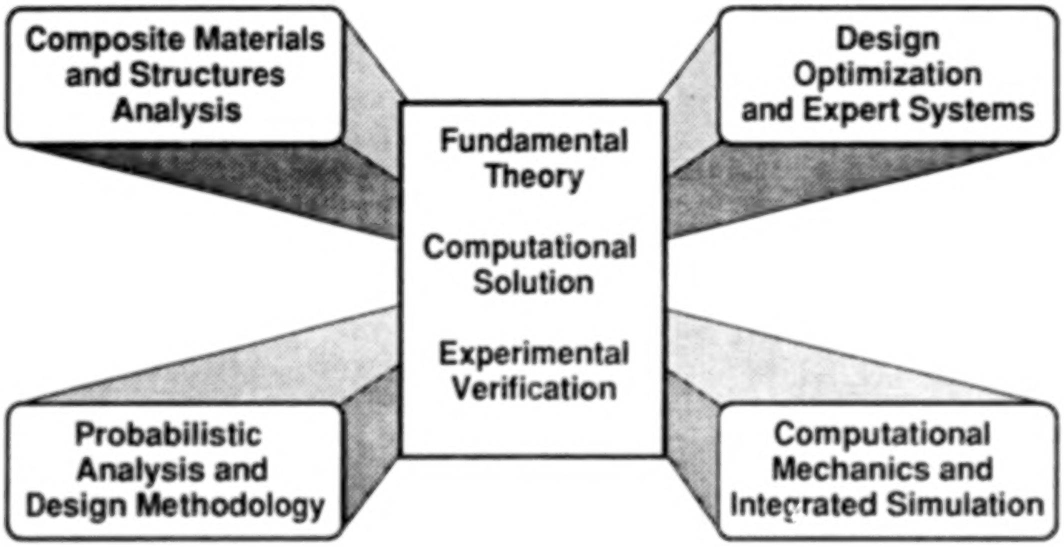
The chronicle of aeropropulsion development reveals a deliberate evolution during which new engine designs have been derived by incremental improvements on successful previous systems. Aerospace vehicles envisioned near the turn of the century and beyond demand advances in propulsion systems of more revolutionary than evolutionary significance. The systems of tomorrow will require unprecedented levels of performance, durability, reliability, and operational economy. Achieving these requirements presents a significant challenge to develop enabling computational structures technology.

An onerous consequence now endured because of deficient analysis and design capabilities is the reliance on hardware tests to demonstrate and certify engine system requirements. Indeed, a considerable amount of the total time and cost associated with developing a new engine can be attributed to the several iterations that typically occur in the design-build-test cycle. Alleviating the dominance of hardware tests can substantially reduce the time and cost of propulsion development. The aim of computational structures technology is to transform the engine development process by empowering computational simulation to have the principal role.

The arena of computational structures technology for aeropropulsion has produced some noteworthy recent gains, and even more extraordinary advances are still to be realized. The essential elements in this endeavor are (1) fundamental theoretical models that more completely represent the complex physics governing engine structural performance, (2) computational techniques that provide accurate and efficient solutions of the governing models and which exploit the potential of emerging computer technology, and (3) integrated strategies for simulation that allow engine structural models of varying fidelity to be evaluated as a continuous and adaptive process.

The obvious benefit of the new capabilities that are being pursued is a greater opportunity to examine alternative concepts or to address other issues that are pertinent to engine system design. More profoundly, the new capabilities can be utilized to address major obstacles that are now only confronted through hardware testing. These efforts promise a new potential that inevitably will stimulate ideas for overcoming future propulsion barriers. Success in these endeavors will contribute to an ideal of "computation to flight" for aerospace propulsion systems. It is this ideal that inspires the research and development of computational structures technology at Lewis Research Center.

Computational Structures Technology Development is Broad Based and Comprehensive



Composite Materials and Structures Analysis

Design Optimization and Expert Systems

Fundamental Theory

Computational Solution

Experimental Verification

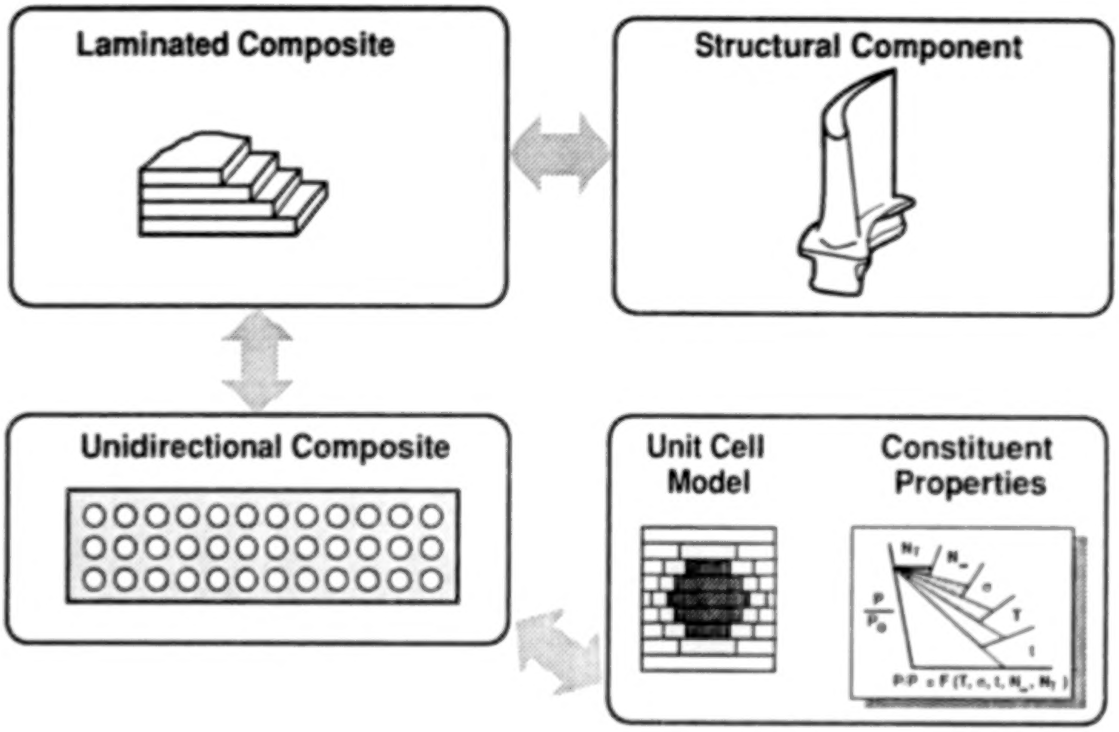
Probabilistic Analysis and Design Methodology

Computational Mechanics and Integrated Simulation

CD-91-56424

This presentation gives a partial overview of research and development efforts underway in the Structures Division of Lewis Research Center, which collectively can be referred to as the computational structures technology program. The efforts in the program are diverse and encompass four major categories: (1) composite materials and structures, (2) probabilistic analysis and design methods, (3) design optimization and expert systems, and (4) computational methods and integrated simulation. The approach of the program is comprehensive and entails (1) exploration of fundamental theoretical models of structural mechanics that more completely represent the complex physics governing engine structural performance; (2) formulation and implementation of computational techniques and integrated simulation strategies that provide accurate and efficient solutions of the governing theoretical models by exploiting the potential of emerging computer technology, and which allow engine structural models of varying fidelity to be evaluated with arbitrarily specified resolution as a continuous and adaptive process; and (3) validation and verification through numerical and experimental tests to establish confidence and define the qualities and limitations of the resulting theoretical models and computational solution methods. The program comprises both in-house and sponsored research activities. The remaining pages provide a sampling of the activities to illustrate the breadth and depth of the program and to demonstrate the accomplishments and benefits of the program.

Multiscale Approach Relates Local Effects to Global Behavior of Composite Structures


Laminated Composite

Structural Component

Unidirectional Composite

Unit Cell Model

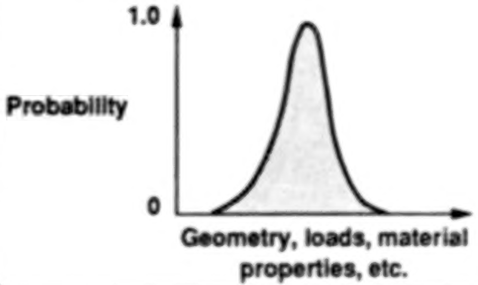
Constituent Properties

CD-91-54425

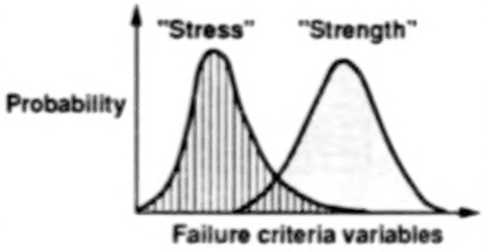
The thermomechanical performance and structural integrity of intermetallic and ceramic matrix composites in high-temperature engine applications are governed by the behavior of the constituent materials. Local influences which can affect the behavior of a structural component are caused by factors such as imperfect bonding and slipping at the fiber-matrix interface, a progressive damage and failure process induced by microcrack development in the typically brittle matrix materials, and constituent material properties which vary nonlinearly and may exhibit a cyclic history dependence over the range of operating conditions experienced by candidate engine components. For the analysis and design of engine components fabricated of these materials, it is desirable to account for the local factors and to relate their effects on the global structural performance. For this purpose, an integrated multiscale approach has been developed, as depicted in the figure. In the tradition of earlier approaches for polymeric and metallic matrix composites, the new capability incorporates (1) nonlinear constituent material models and failure models, (2) composite micromechanics and macromechanics models, and (3) finite-element global structural analysis models. The unique unit cell which serves as the basis of the micromechanics model allows an arbitrary level of resolution of locally nonuniform or discontinuous behavior of material properties, stress and strain, temperature, and other critical variables. Despite this capability for capturing local detail, the multiscale approach maintains a practical computational efficiency for realistic engine component analyses.

Probabilistic Analysis Methods Quantify Risk for Better Design Decisions

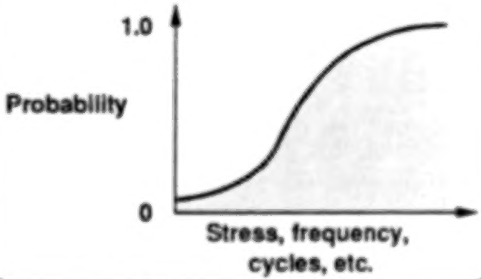
Parameter Uncertainty



Risk Assessment



Behavior Likelihood



Risk Management

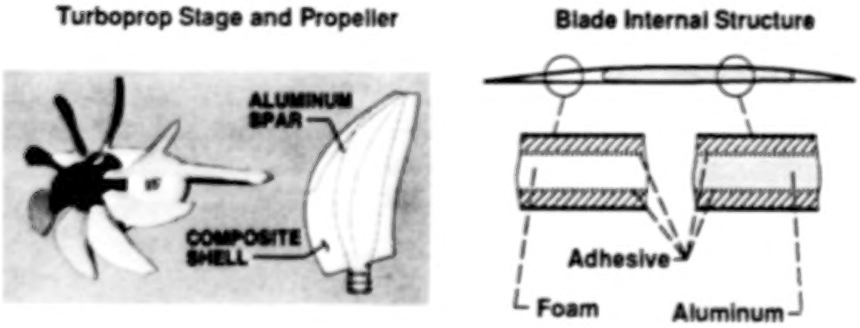
- Manufacturing quality control
- Inspection interval
- Maintenance schedule
- Qualification tests
- Flight certification limits

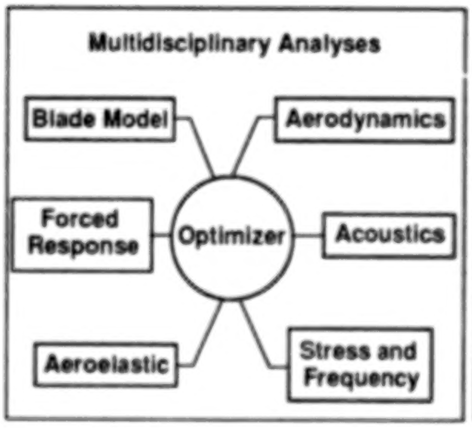
CD-91-54426

The variables of the structural design process, including geometry, material properties, loads, and boundary conditions, exist only with a certain degree of natural variability. The uncertainty of these variables contributes to an associated level of risk that the resulting design for a structural component will perform unexpectedly. The level of risk in a design is not directly quantified by the traditional deterministic design methodology. Rather, the deterministic design methodology devised the concept of the "safety factor" as a qualitative indicator of the risk in a design. This approach is inherently conservative and provides no basis to attain the desired balance between safety and efficiency of the design. A probabilistic analysis and design methodology, on the other hand, provides the formalism to quantify design uncertainty. As the figure implies, a probabilistic methodology establishes a rational basis for assessing risk and making risk management decisions. A probabilistic analysis and design methodology is especially pertinent in the design of high-performance, high energy-propulsion systems where mission economy and safety are the primary (and competing) design objectives. In this case, the ability to accurately quantify reliability and risk is essential to achieve an acceptable balance between performance and safety.

Structural Tailoring Resolves Complex Requirements for Optimum Design

Advanced Propfans: Complex Geometry and Construction





Multidisciplinary Analyses

Blade Model

Aerodynamics

Forced Response

Optimizer

Acoustics

Aeroelastic

Stress and Frequency

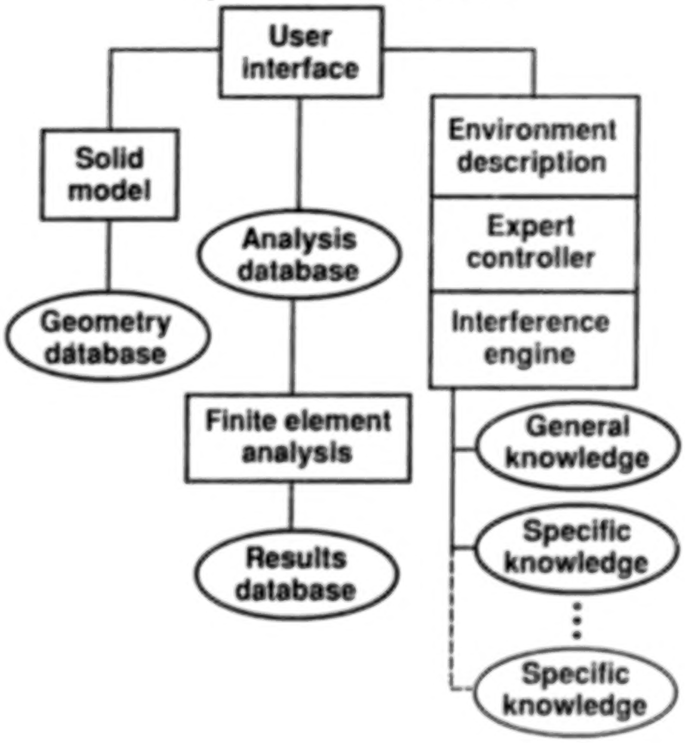
Propfan SR7 Optimization

	Initial design	Final design
Efficiency, %	82.86	83.17
Noise, dB	143.8	137.3
Weight, lb	41.1	41.2
Cost reduction, %		3.32

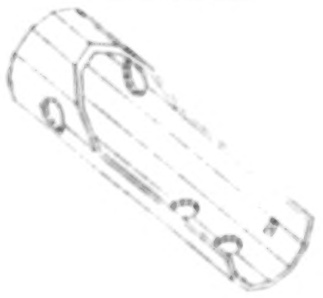
The traditional approach to engine component design has been to satisfy competing multidisciplinary requirements independently through manual design iterations. This process, which is usually conducted between several discipline-specific groups, inherently becomes time consuming and expensive, cumbersome and error-prone, and highly subjective. The typical process, therefore, can be carried out only to the point where a satisfactory design is achieved. The luxury of continuing the process to find the best design is virtually never afforded. Particularly relevant examples of this design scenario occur in the case of engine blades. Recent development of the advanced turboprop propulsion concept presents a consummate example of the ponderous task of trying to satisfy multidisciplinary design requirements. The design of advanced propfan blades presented the opportunity to demonstrate an alternative strategy. The approach taken was to streamline, automate, and formalize the propfan design process by incorporating the multiple discipline-specific analyses together with numerical optimization techniques into a computationally effective design-tailoring system. As summarized in the figure, the design-tailoring strategy proved to be highly successful for the propfan application. The concept of component-specific design tailoring has been successfully extended to cooled turbine blade applications and, most recently, a cooled wall panel structure for hypersonic engine inlets.

Expert Systems Capture Heuristic Knowledge To Guide Structural Modeling and Analysis

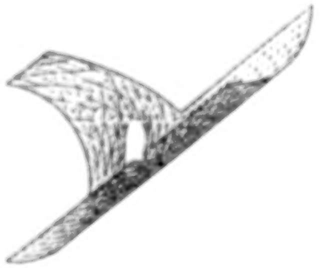
System Architecture


User interface
Solid model
Geometry database
Analysis database
Finite element analysis
Results database
Environment description
Expert controller
Interference engine
General knowledge
Specific knowledge
Specific knowledge

Solid model



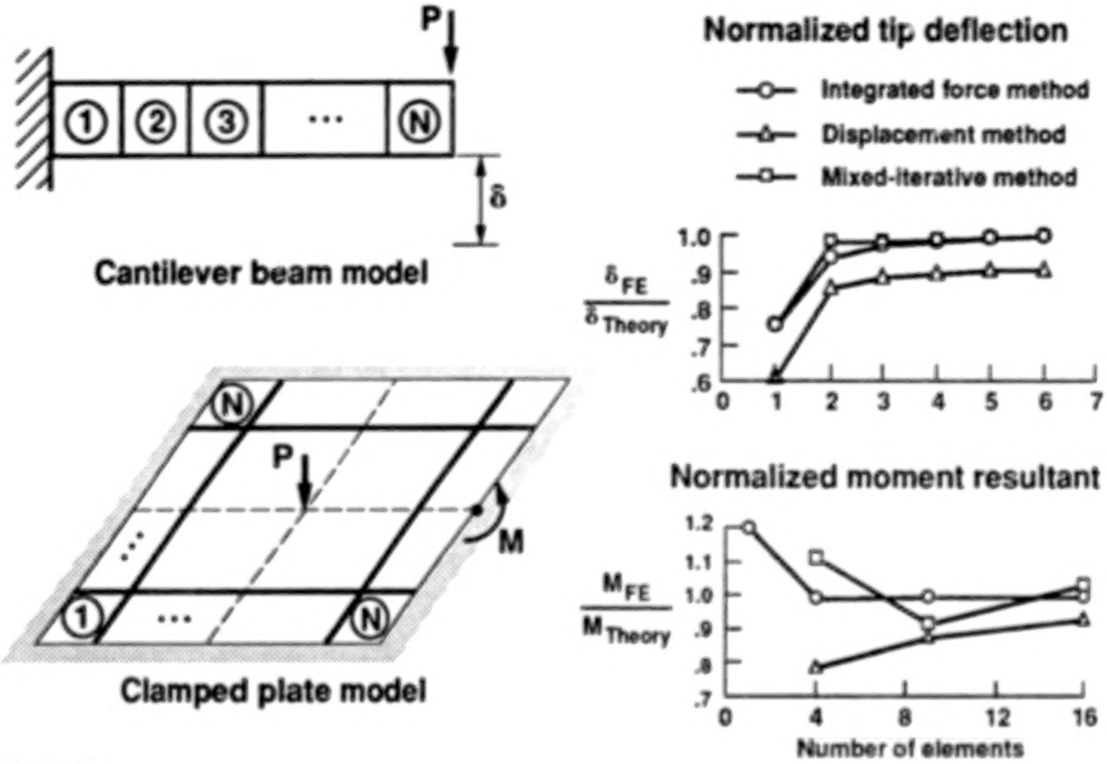
Discrete model



CD-91-54429

The creation of geometric and discrete models of structural components for analysis by the predominant finite-element method remains a subjective process that relies to a great extent on the experience and judgement of the structural engineer. Of considerable interest is the notion of capturing the heuristic reasoning and knowledge that constitutes the structural design process. The potential benefit of such a concept is to enable less experienced engineers to consistently create more effective models and achieve more reliable analyses. The ability to configure an "advisor" for structural modeling and analysis has been demonstrated recently with the development of the automated design expert (ADEPT) system. The ADEPT system combines solid- and discrete-model-creation facilities with an expert system that embodies knowledge pertaining to the assumptions and methodology of finite-element structural analysis. The ADEPT system guides the engineer through an examination of various features of the component model including geometric attributes and loading and boundary conditions. The ADEPT system makes recommendations for creating the appropriate and most effective discrete model for subsequent solution using specific finite-element analysis application programs. As the figure illustrates, the feasibility of a specialized expert system such as ADEPT for assisting engineers in the structural modeling and analysis process has been demonstrated for complex configurations typical of engine components.

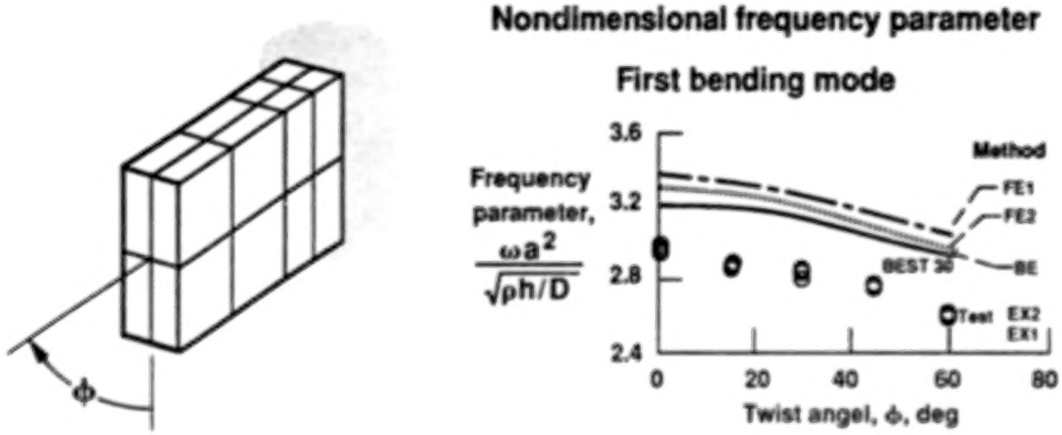
Advanced Finite-Element Methods Provide More Effective Structural Analyses



CD-81-64430

Reliable and cost-effective engine component design requires accurate and efficient structural analysis tools. The finite-element method is clearly the predominant tool utilized today for the analysis of complex structures in general. Furthermore, the overwhelming majority of production-level, finite-element-application programs are based on the conventional "displacement" or "stiffness" method concept which originated over thirty years ago. Despite this prominence, the displacement-based, finite-element method exhibits deficiencies particularly in its ability to resolve internal force and stress fields. These limitations generally require the utilization of very dense finite element models with many degrees-of-freedom to adequately resolve the important field quantities. The penalty associated with this is manifested in both the person-time to create the model, the computational resources to conduct the analysis, and the difficulty of assessing the quality of the analysis results. Two new formulations for finite-element analysis are being developed (the mixed-iterative method and the integrated-force method) in the attempt to alleviate the shortcomings of the displacement method. The results presented in the figure are for simple test cases that have recently been investigated to examine the potential benefits of the new methods. As seen in the results, the new methods appear to provide more accurate representations of both displacement and stress fields with sparser models. This advantage is more significant for the larger models of real engine components.

Advanced Boundary–Element Methods Provide a Viable Alternative For Structural Analyses



Nondimensional frequency parameter

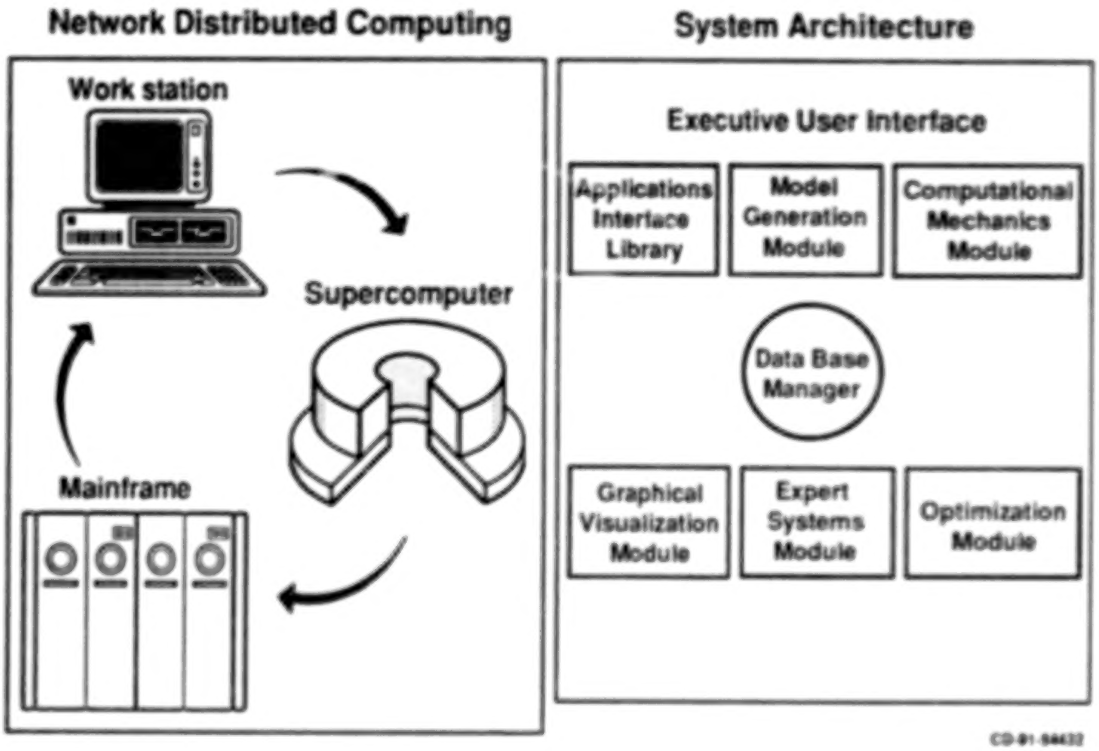
First bending mode

Result designator	Description
FE1	Finite-element shell model (294 DOF)
FE2	Finite-element solid model (660 DOF)
BE	Boundary-element model (288 DOF)
EX1	NASA experimental data
EX2	AF experimental data

CD-81-54431

Although the boundary-element method has been recognized for nearly as long as the more prominent finite-element method, it has been given negligible development effort and, therefore, has had limited utility as a structural analysis tool. The fundamental advantage that the boundary-element method possesses is that, for certain types of problems, it requires the discretization of only the bounding surface of a structure and not the complete domain. This fundamental advantage, unfortunately, extended only to a very restricted class of problems for which the fundamental integral representation contained no domain components. It is because of the early perception that boundary-element methods would never have as general a utility as finite-element methods that it received such scanty attention until only recently. Concerted efforts have been made in the last several years to extend the viability of the boundary-element method to a much broader range of problems. The focus of these efforts has been to contend with the special circumstances faced in engine component analyses, including complex geometry, anisotropic materials with history-dependent inelastic behavior, cyclic loading, and heterogeneous boundary-conditions, etc. These efforts have culminated in a comprehensive and general-purpose boundary-element structural analysis tool. The capabilities provided by this tool give the engineer a significant new alternative for structural analysis to supplement the existing arsenal of finite-element tools. As illustrated by the example results given in the figure, the boundary-element method has gained a new superiority for solving a broader range of engine structural analysis problems.

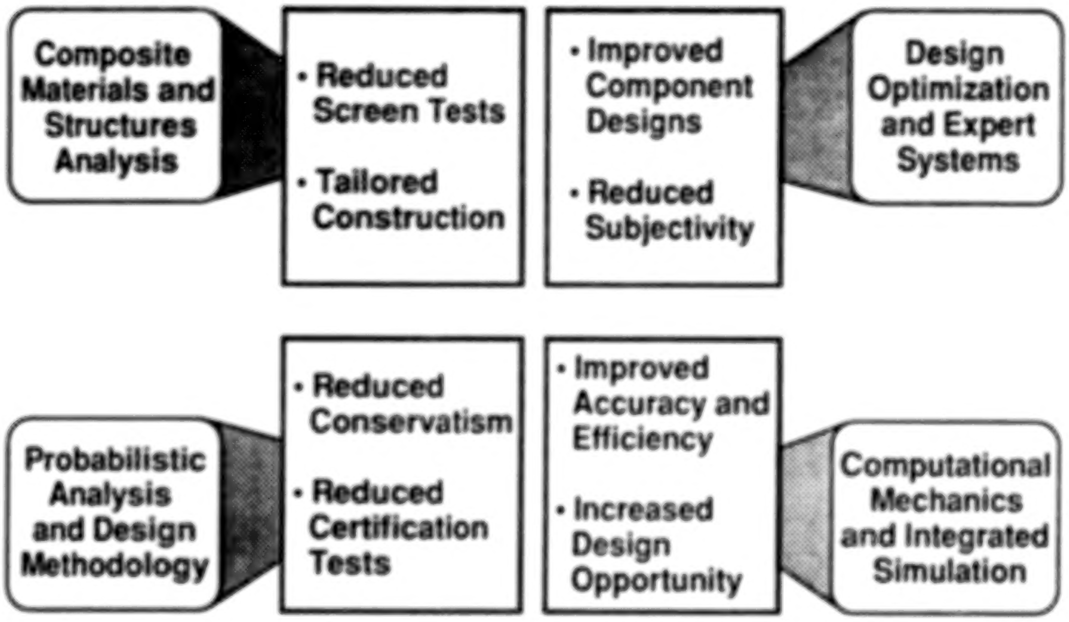
The Engine Structures Simulator—A Unified System For Adaptive Analysis and Design


Network Distributed Computing
Work station
Supercomputer
Mainframe
System Architecture
Executive User Interface
Applications Interface Library
Model Generation Module
Computational Mechanics Module
Data Base Manager
Graphical Visualization Module
Expert Systems Module
Optimization Module

CD-91-54432

The Engine Structures Computational Simulation project has the objective to investigate, develop, and implement the required technologies to enable the simulation and synthesis of engine systems structural performance. *Engine structure* is used in the general context and refers to any extent of the engine which is of interest. This can range from a subcomponent (airfoil) to a component (blade), an assembly (rotor), a subsystem (compressor), or the complete system (turbofan engine). *Structural performance* refers to any behavior parameter or measure of merit such as stress or strain, vibration frequency, life expectancy, weight, reliability level, or cost. The concept herein of *simulation* implies a continuous process that permits creation and evaluation of engine structural performance models as an adaptive and continuous process. The simulation process should permit simultaneous models of varying fidelity and solution of those models with arbitrarily specified resolution. The principle end product of the project is a computational system (Simulator) which embodies the strategy and capacity to perform the diverse tasks that comprise the simulation process. This *computational system* represents the synergism of computational mechanics methodology and computer software and hardware implementation. It follows that the Simulator is a comprehensive entity encompassing aspects of model generation, computational mechanics, information management, graphical visualization, simulation process control, software system engineering, distributed computing, and so on. Providing the engine structural simulation capability envisioned demands unprecedented utility, flexibility, and adaptability within a unified framework.

Computational Structures Technology Can Significantly Enhance Future Propulsion Systems



Composite Materials and Structures Analysis

- Reduced Screen Tests
- Tailored Construction

- Improved Component Designs
- Reduced Subjectivity

Design Optimization and Expert Systems

Probabilistic Analysis and Design Methodology

- Reduced Conservatism
- Reduced Certification Tests

- Improved Accuracy and Efficiency
- Increased Design Opportunity

Computational Mechanics and Integrated Simulation

CD-91-54433

This presentation gives a partial overview of research underway in the Structures Division of Lewis Research Center, which collectively can be referred to as the computational structures technology program. The accomplishments anticipated in this program will generally expand the benefits of computational simulation far beyond the capability of today's analysis and design practice. To be more specific, some expected benefits from each of the four major categories discussed are

(1) *composite materials and structures* - reduced requirements for candidate composite material screening tests and new opportunities for tailored material and structural design

(2) *probabilistic analysis and design methods* - reduced design conservatism and reduced requirements for hardware certification tests

(3) *design optimization and expert systems* - improved component designs and reduced subjectivity of the design process with more consistent success

(4) *computational methods and integrated simulation* - improved accuracy and efficiency for structural analysis and expanded design opportunities to examine alternative concepts and to address issues that have previously been confronted only through expensive and time-consuming hardware tests

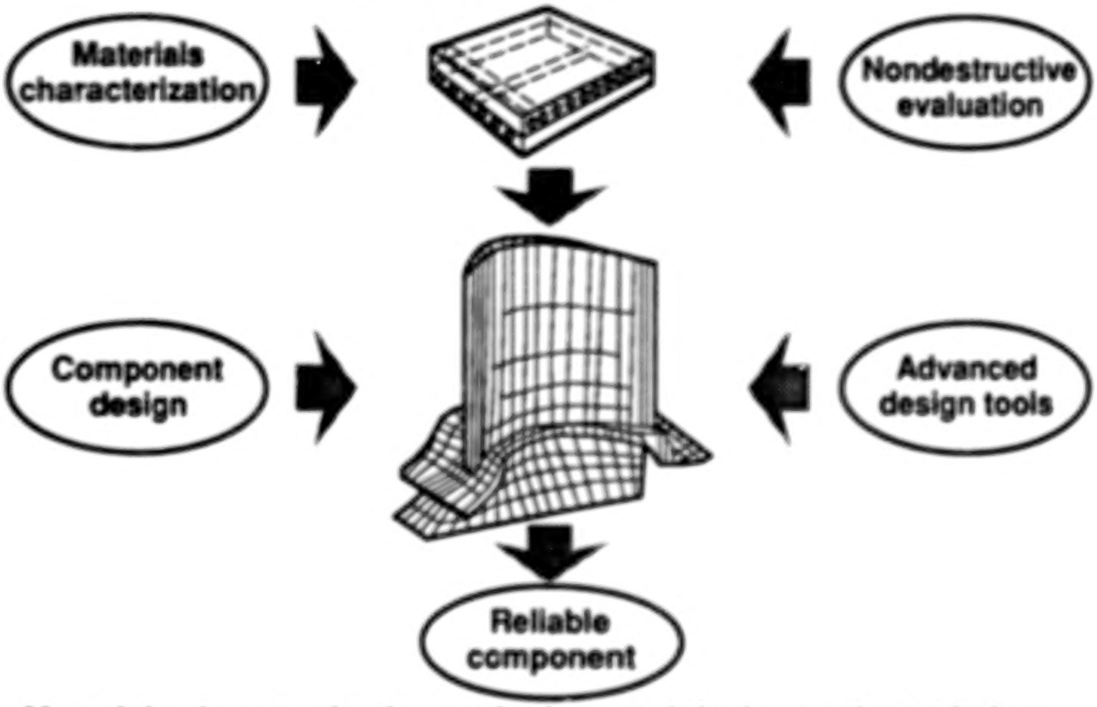
N91-20107

TRANSITION-TO-PRACTICE TECHNOLOGIES FOR BRITTLE MATERIALS

Edward R. Generazio
NASA Lewis Research Center
Cleveland, Ohio

Four major technologies are needed to bring brittle ceramic composites up to the level necessary for practical application as aerospace materials. These are mechanical testing, nondestructive evaluation (NDE), mechanical design, and life prediction. An advanced design tool, C-CARES, has been developed to assist designers in the mechanical design of structural ceramic components by determining the reliabilities of these components. In situ and in process NDE are being used to enhance/accelerate development of CMC's. As a very minimum both x-ray and ultrasonic imaging needs to be performed for an accurate NDE. In situ and in process NDE also provides validation of analytical models.

Transition-to-Practice Technologies for Brittle Materials



Materials characterization

Nondestructive evaluation

Component design

Advanced design tools

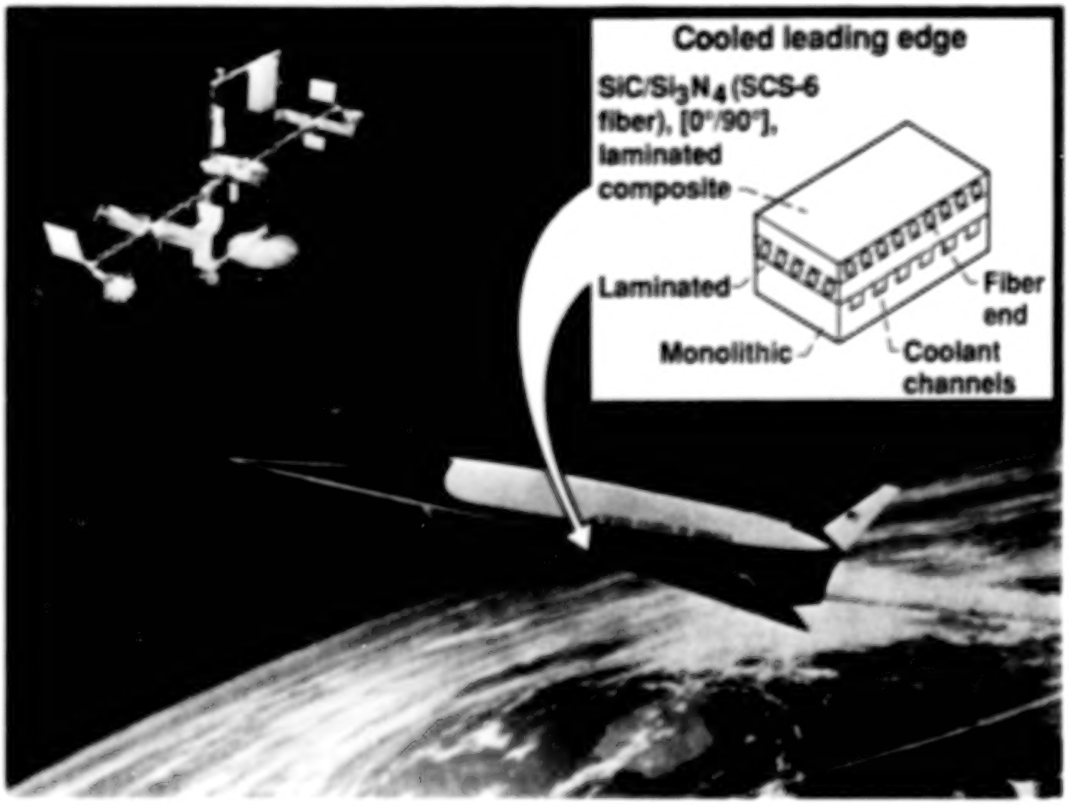
Reliable component

- **Materials characterization and advanced design tools are being used to develop ceramic components with high reliabilities.**

CD-91-[illegible]

Transition-to-practice technologies are needed to bring brittle ceramic composites up to the level necessary for practical application as aerospace materials. Two key areas where technologies are being used to develop ceramic composites are Materials Characterization and Component Design. Materials characterization techniques, both destructive and nondestructive, yield information on the material's failure mechanisms, integrity and degradation. This crucial information guides and enhances the development of ceramic composites. Advanced design tools are also being developed to assist component designers with structural ceramic composite component development. These tools strongly influence the design of structural ceramic composite components by determining the component's reliability and life expectancy.

Transition–to–Practice Technologies for Brittle Materials


Cooled leading edge
SiC/Si$_3$N$_4$ (SCS-6 fiber), [0°/90°], laminated composite
Laminated
Monolithic
Fiber end
Coolant channels

CD-91-53687

Advanced high-temperature, low-density composite materials are being developed for use in the next generation of aerospace systems. The High Speed Civil Transport (HSCT), an aircraft for transporting 250 passengers at Mach 3.2 for 5000 n mi and the National Aerospace Plane (NASP), a transportation system that will take off, fly at Mach 25 directly into orbit and land like a conventional aircraft, will require advanced composite materials for both propulsion and structural components.

Transition–to–Practice Technologies for Brittle Materials

- **Mechanical testing**
- **Nondestructive evaluation (NDE)**
- **Mechanical design**
- **Life prediction**

CD-91-[illegible]

The four major transition-to-practice technologies are needed to bring composite materials from the current developmental to the practical applications stage needed to develop the HSCT and NASP. The four areas are mechanical testing, nondestructive testing, mechanical design, and life prediction. Mechanical and nondestructive testing are technologies needed for materials characterization. Mechanical design and life prediction are technologies that make use of the materials properties in order to produce reliable, long life components.

Nondestructive Evaluation for Component Development

- In process NDE- characterizes microstructure and flaw populations that affect strength and life.
- In Situ NDE- Performed during testing identifies occurrences of failure mechanisms.
- Both provide feedback for materials development and for validating analytical models.

CD-91-53893

Two types of nondestructive evaluation (NDE) are needed for developing ceramic composite components: in situ and in process NDE. In process NDE is done during the development of materials to characterize microstructural and flaw populations that affect strength and life. In situ NDE is performed during testing and is used for identifying the occurrence of failure and the failure mechanism. Both types of NDE provide feedback for materials development and for validating analytical models.

Nondestructive Evaluation Tools for Materials Development

- X-ray radiography
- Ultrasonic imaging
- Acoustic emission

CD-91-53894

Many tools are available for performing nondestructive evaluation. The three most important for characterizing ceramic composites are acoustic emission, x-ray radiography, and ultrasonic imaging. Acoustic emissions are ultrasonic signals generated during the actual fracturing of fibers or matrix material. X-ray radiography and ultrasonic imaging are similar to their medical equivalents of x-ray and ultrasound.

In Process NDE Objective

Nondestructively evaluate ceramic composites for microstructure and flaw population that affect strength and life.

- **Cracks**
- **Fiber alignment**
- **Bond quality**
- **Porosity**

CD-91-53887

The objective of in process NDE is to nondestructively evaluate ceramic composites for microstructure and flaw populations that affect strength and life. The most important are cracks, fiber alignment, bond quality variations, and porosity.

Nondestructive Assessment of Impact Damage in CMC Composites

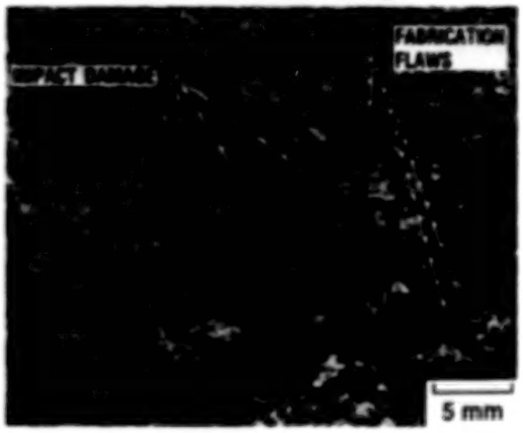
SiC_F/Si_3N_4 Ceramic Matrix Composite Laminate

X-ray radiograph

Before impact

After impact

Ultrasonic scan image

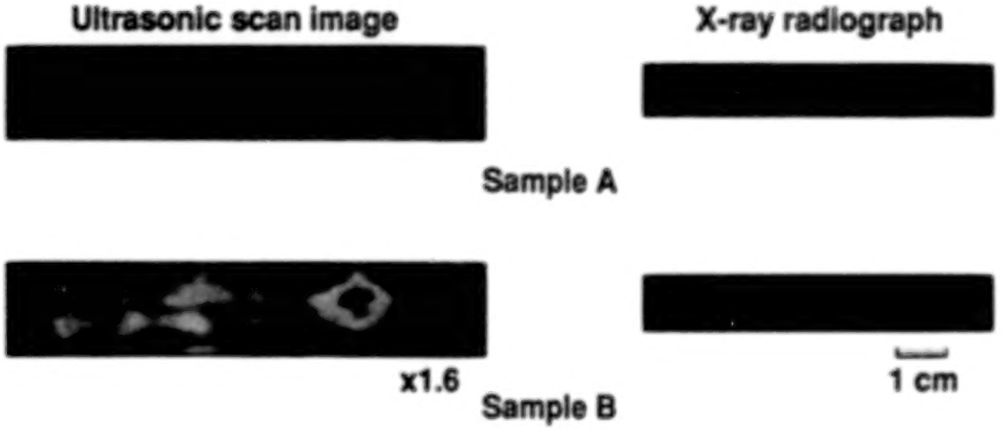


Comparison of radiographs and ultrasonic images shows cracks pre-existing the impact damage, which is confined to delaminations near the impact zone.

CD-91-53898

A SiC fiber/Si_3N_4 composite panel was impacted with a projectile. Before and after x-ray radiographs show cracks pre-existing. The impact damage is confined to the delaminations in the vicinity of the impact zone.

Nondestructive Evaluation of Hot-Pressed SiC Fiber/Si_3N_4 Laminated Composite

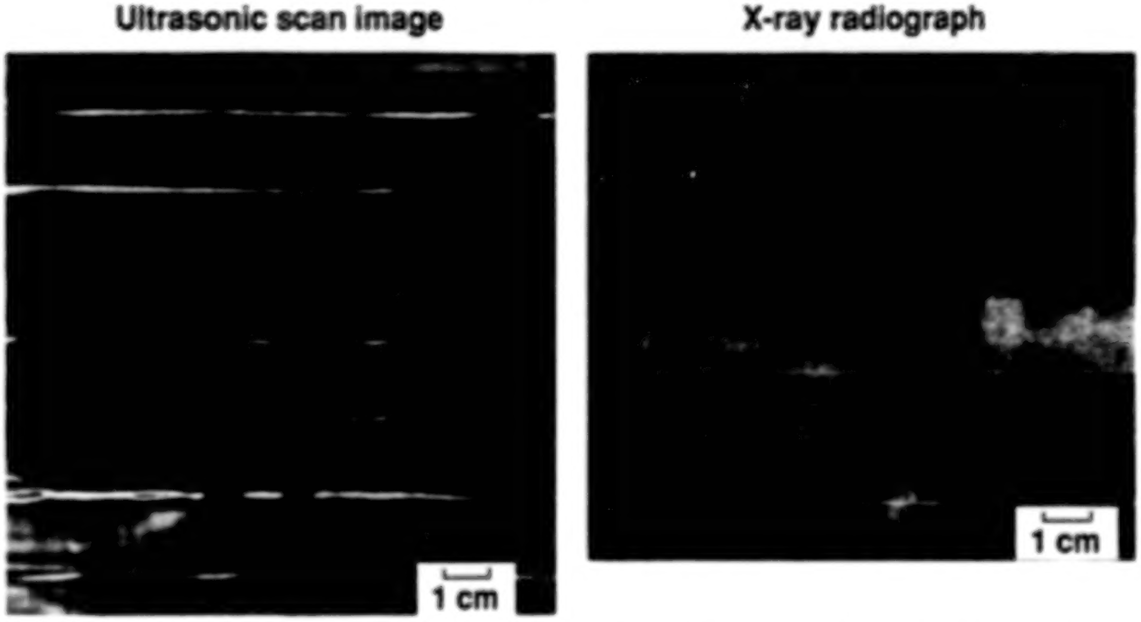


- X-ray images indicate that identically produced samples have similar density structures.
- Light areas in acoustic image identify regions exhibiting "good" bond between lamina.

CD-91-53899

Two similarly produced SiC fiber/Si_3N_4, laminated composite samples have essentially identical x-ray radiographic results. However, the ultrasonic images indicated that the bonding between the lamina is quite different for the two samples. A laminated composite sample that has a good bonding between layers will have strong ultrasonic through transmission signals, which will appear as lighter areas in the image. Sample B has higher (brighter) ultrasonic transmission properties than sample A; therefore, sample B has "good or better" bond between layers.

Nondestructive Evaluation of Nicalon/CAS [0°/90°] Laminated Composite


Ultrasonic scan image

X-ray radiograph

- Ultrasonic and X-ray images reveal low density regions after hot pressing.
- Both images reveal similar features—no debonds or delaminations are present.

CAS = Calcium aluminosilicate glass

CD-91-53900

X-ray and ultrasonic analysis may yield identical results. The dark areas of the ultrasonic image correspond with the areas of low density (high porosity) in the x-ray radiograph.

ORIGINAL PAGE IS
OF POOR QUALITY

Nondestructive Testing of Laminated Ceramic Composite Panels

X-ray image

Ultrasonic image

- X-ray image reveals misaligned/bowed fibers.
- Ultrasonic image reveals isolated delaminated (dark) regions.
- Images obtained before and after testing provide crucial information on failure mechanisms.

CD-91-53901

The x-ray radiograph of a laminated SiC fiber/Si_3N_4 composite panel indicates that the fibers are misaligned or bowed. The ultrasonic image reveals dark regions where there is poor bonding between lamina.

ORIGINAL PAGE IS
OF POOR QUALITY

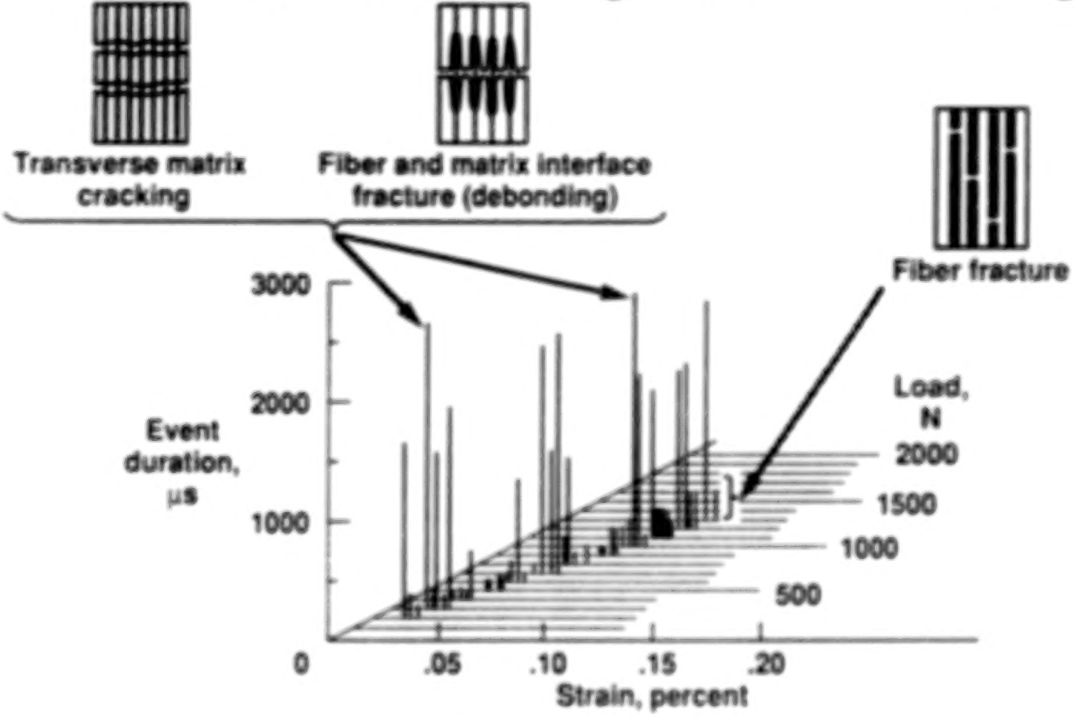
In Situ NDE Objective

- Identify the occurrence of failure mechanisms during loading.
  - Transverse matrix cracking
  - Fiber fracture
  - Fiber interface failure

CD-91-53895

One objective of in situ NDE is to identify the occurrence of failure mechanisms during loading. These mechanisms are transverse matrix cracking, fiber fracture, and fiber interface failure.

Acoustic Emissions During Mechanical Testing



- **Identifies damage mechanisms.**
- **Validates analytical models.**
- **Assists material development**

CD-91-[illegible]

During loading fracturing fibers and matrix material generate acoustic signals that may be grouped according to the duration of the signal. Acoustic emissions that have a long duration are due to transverse matrix cracking or fiber/matrix interface debond. Short duration acoustic signals are indications of individual fiber fracture. The occurrence of damage mechanisms is used to validate models and assist in the material development.

Ceramic Matrix Composite Design Analysis Objective

Develop and refine analytical methods and computer codes for predicting

- **Fast fracture**
- **Component life**

for laminated ceramic composites used as structural components in aerospace applications that entail high-temperature environments.

CD-91-53889

Ceramic materials are brittle. The mechanical failure of these materials follows probabilistic failure theories. Therefore, probabilistic methods need to be incorporated in the design when developing a specific component. Analytical methods and computer codes are being developed for predicting fast fracture failure and life remaining of laminated ceramic composite components.

Computer Program Developed

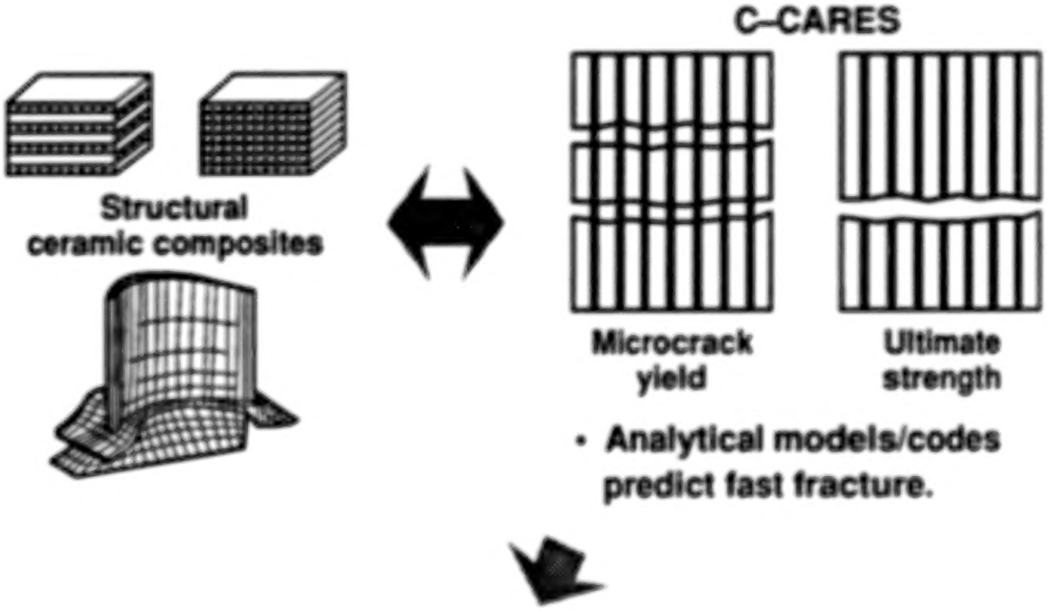
C-CARES:

Composite Ceramic Analysis and Reliability Evaluation of Structures

CD-91-53890

A computer program has been developed to predict fast fracture of ceramic composite components. The program is available as Composite Ceramic Analysis and Reliability Evaluation of Structures or C-CARES.

C–CARES: A Needed Design Tool


Structural ceramic composites
C–CARES
Microcrack yield
Ultimate strength

- Analytical models/codes predict fast fracture.

- Component reliability
- Component life

C-CARES identifies the microcrack yield and ultimate strength of structural ceramic composites. Since C-CARES is based on probabilistic failure theories that represent these advanced ceramic composites, an overall component reliable is determined.

Status of and Plans For C-CARES Code

- First version coupled to MSC/NASTRAN.
- Ply level analysis included.
- Three modes of failure: transverse to fiber, parallel to fiber, and in-plane shear.
- Extend theory to include time dependent mechanisms that determine component life.

CD-91-[illegible]

The first version of the C-CARES computer code has been coupled to a finite-element analysis program, MSC/NASTRAN. The stress states of components determined from MSC/NASTRAN are used directly by C-CARES. C-CARES analysis can be done at the ply level and permits three modes of failure: transverse to the fiber, parallel to the fiber, and in-plane shear. The C-CARES code is being extended to include time dependent mechanisms that determine component life.

CONCLUSION

The four major technologies needed to bring ceramic composites up to a level for practical application as aerospace materials are: mechanical testing, nondestructive evaluation, mechanical design, and life prediction. An advanced design tool, C-CARES, has been developed to assist designers in the mechanical design of structural ceramic components by determining the reliabilities of these components. In situ and in process NDE are being used to enhance/accelerate development of CMC's. As a very minimum both x-ray and ultrasonic imaging needs to be performed for an accurate NDE. In situ and in process NDE also provide validation of analytical models.

- **Mechanical design**
- **Nondestructive evaluation**
- **Mechanical testing**
- **Life prediction**

Are technologies needed to develop CMC's for practical aerospace applications?

Advanced design tool, C-CARES, may be used to determine reliabilities of structural ceramic components.

In Situ and in process NDE enhance/accelerate development of CMC's, and provide validation of analytical models.

CD-91-53902

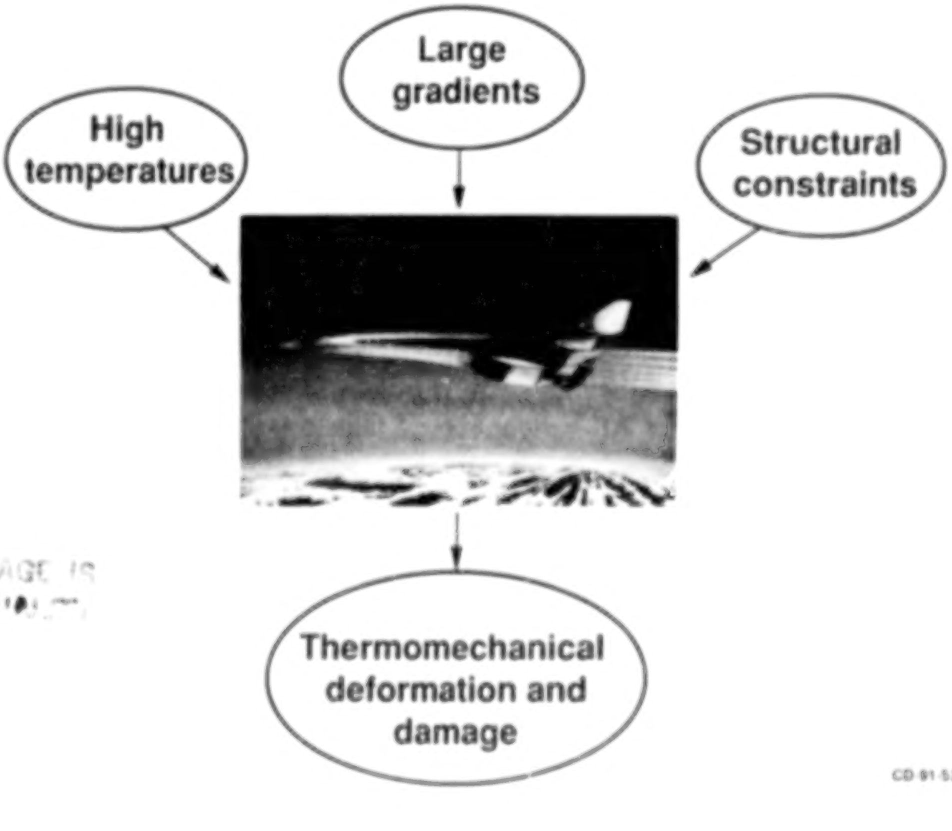
N91-20108

PROGRESS IN MODELING DEFORMATION AND DAMAGE

Alan D. Freed and Steven M. Arnold
NASA Lewis Research Center
Cleveland, Ohio

In hypersonic aircraft, for example, high-temperature structures (such as leading edges, inlet cowls, combustor liners, and nozzles) are subjected to thermomechanical deformations. It is believed that such deformations will be a primary cause of structural failure in these components. They arise from large thermal gradients across structural skins that are constrained from free thermal expansion. In order to assess structural life and performance, the material's thermomechanical behavior must be incorporated into the structural design. The laboratories of the Fatigue and Fracture Branch are dedicated to observing the evolution of deformation and damage in laboratory specimens under thermomechanical loading conditions. These observations are used within the Branch to guide the development of deformation and life-assessing models.

Problem Statement



High temperatures

Large gradients

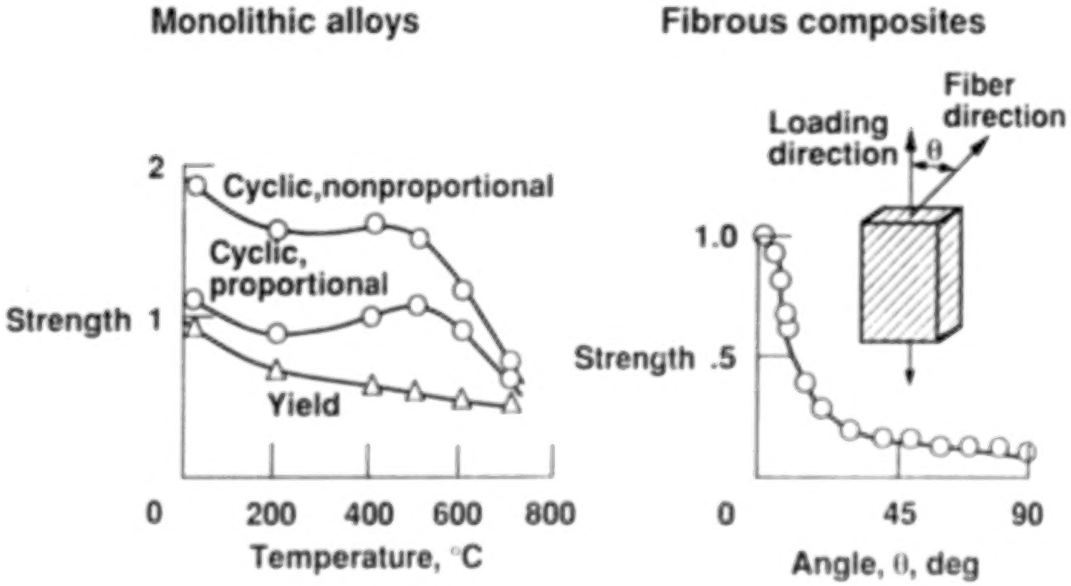
Structural constraints

Thermomechanical deformation and damage

ORIGINAL PAGE IS OF POOR QUALITY

CD 91-53904

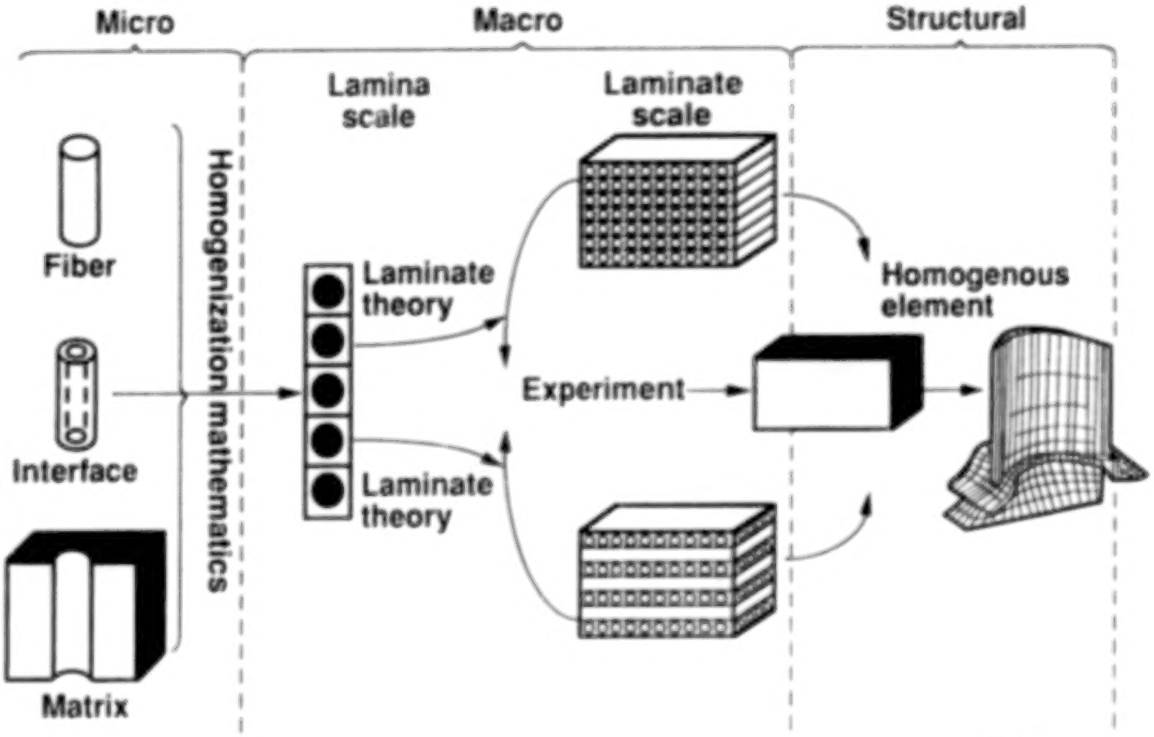
Material Characterization



CD 91 53907

Viscoplastic monolithics and composites both offer many challenges to the viscoplastic model developer. The development of realistic models requires experiments that probe a material's response for a wide variety of inputs. One facet of monolithic material behavior is how temperature and loading history interact to affect material strength. Many alloys gain strength over their original yield strength when they are cyclically loaded. Furthermore, strengthening is often greater in nonproportional cyclic histories than in proportional ones. Also strength does not decay monotonically with increasing temperature. An important feature of composite material behavior is the anisotropy in the structure's strength. For a unidirectional fibrous composite the yield strength falls off rapidly as the axis of loading deviates from that of the fibers in the composite. The loading cone must have a small angle for this type of composite if the designer is to make maximum use of the composite's strength.

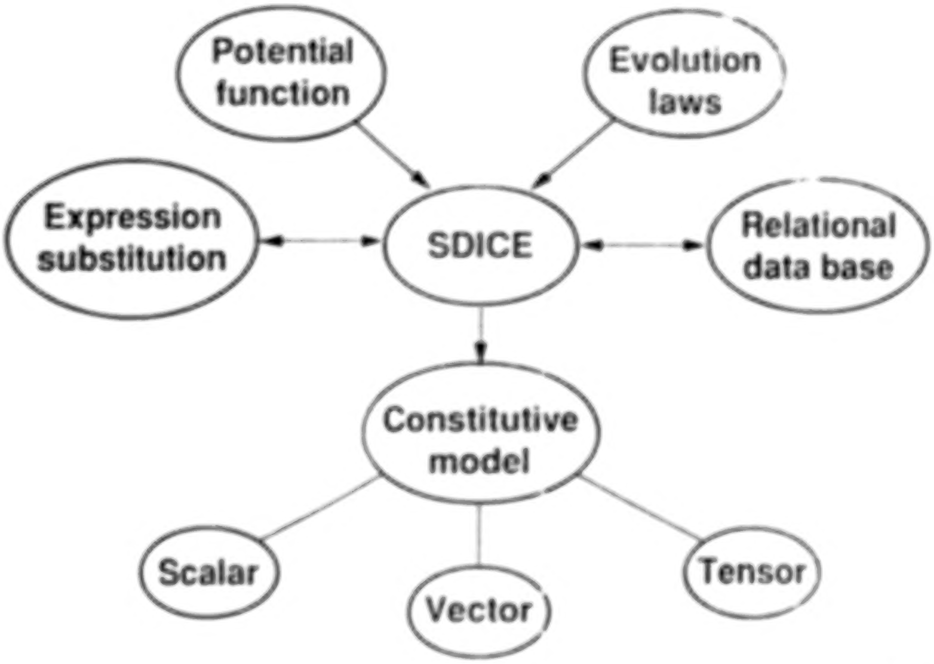
Scales in Composite Analyses


Micro
Macro
Structural
Lamina scale
Laminate scale
Fiber
Interface
Matrix
Homogenization mathematics
Laminate theory
Laminate theory
Experiment
Homogenous element

CD-91-53908

NASA Lewis is investigating two approaches for developing a constitutive model of a fibrous composite whose matrix is viscoplastic and whose fiber is elastic. The first is a micromechanics approach, where the constitutive models for each phase of the composite (i.e., fiber, interface, and matrix) are combined by using homogenization techniques. The result is a constitutive model for the composite. In this approach the geometry of the composite is handled through the mathematics of homogenization. The second approach is a macromechanics one, where the composite is treated as an anisotropic continuum. In this approach the geometry of the composite is handled through the choice of invariants used to describe the anisotropy. Here the constitutive model is defined at a larger scale than is the micromechanics model, and consequently it cannot provide stress and strain information at the constituent (phase) level.

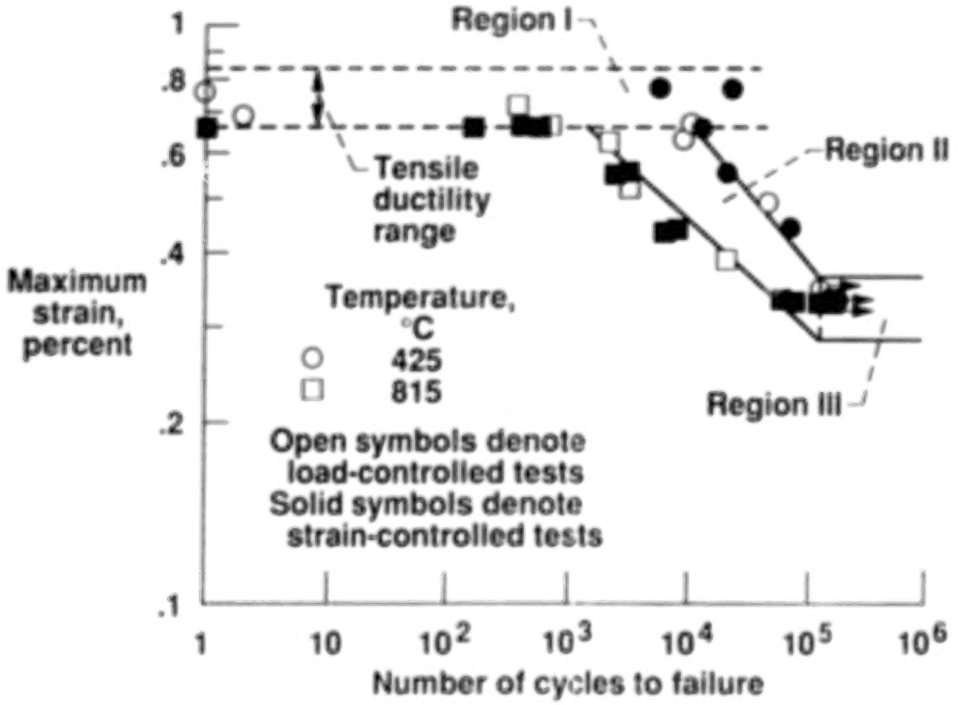
Symbolic Computations



Potential function
Evolution laws
Expression substitution
SDICE
Relational data base
Constitutive model
Scalar
Vector
Tensor

The development of new constitutive equations (e.g., viscoplastic models) can be time consuming, involved, and error prone; therefore, an intelligent application of symbolic systems to facilitate this tedious process can be of significant benefit. A self-contained symbolic expert system, named SDICE, is being developed under NASA Lewis support. This package is capable of efficiently deriving potential-based constitutive models in analytical form. These potential equations can be scalar, vector, or tensor valued. A unique feature of SDICE is its ability to identify common terms found in two or more locations, to assign variables to those terms, and to substitute these variables back into the equations. This important feature minimizes expression growth, which is probably one of the most plaguing problems confronting existing symbolic codes.

Fatigue Life Diagram of MMC's and IMC's



CD-91-53913

Viscoplastic models are defined by a system of mathematically stiff, coupled, nonlinear, first-order, ordinary differential equations. In other words, their numerical integration is "expensive." As a consequence, much research has been expended in the development of efficient and accurate numerical integration methods for solving viscoplastic problems. Probably the most exciting developments in numerical integration theory to come along in many decades are the Walker-Freed asymptotic integration algorithms recently developed at NASA Lewis. For the differential equation

$$\dot{X} + UX = V$$

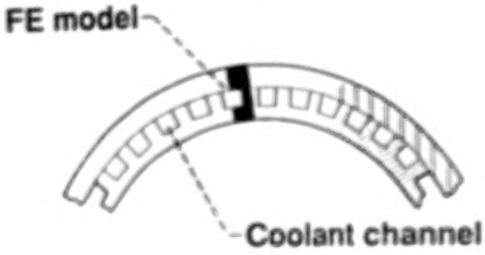
the linear, asymptotic, recursive, integration algorithm is given by

$$X(t + \Delta t) = X(t)e^{-U(t + \Delta t)\Delta t} + V(t + \Delta t)\left[\frac{1 - e^{-U(t + \Delta t)\Delta t}}{U(t + \Delta t)}\right]$$

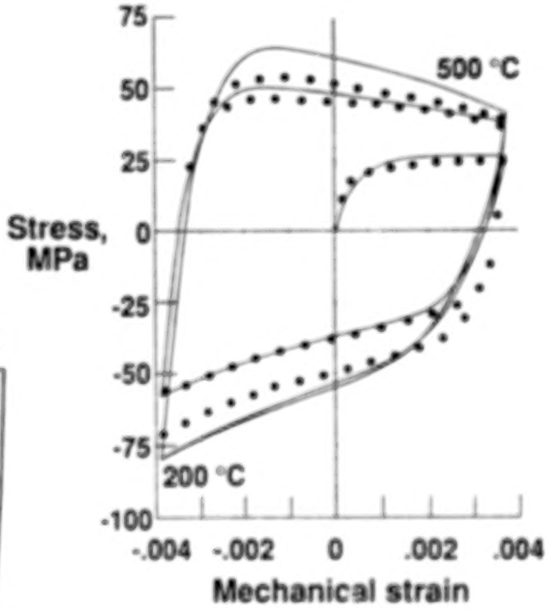
An example of the applicability of this integration algorithm for various time step sizes is shown here. Achieving equal accuracy using the classical method of Runge-Kutta (well suited for stiff differential equations) would require on the order of 10 000 time steps.

Finite Element Analysis

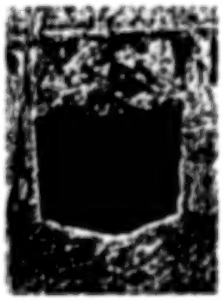
Nozzle liner



Theory vs experiment

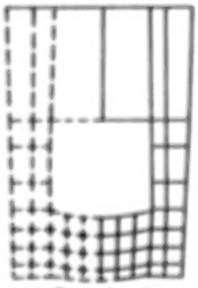


Experiment



Cycle 393

Prediction

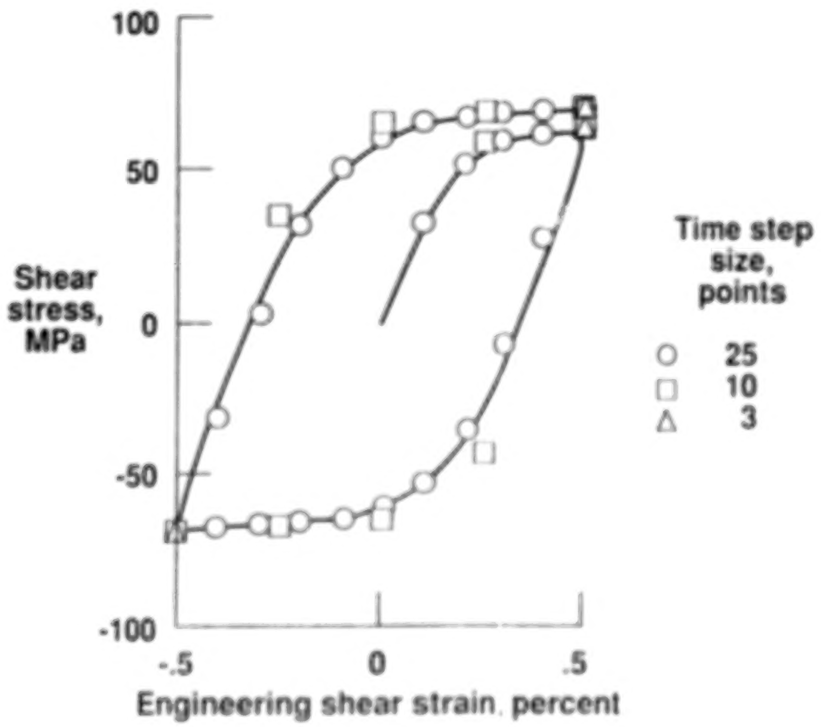


Cycle 6
(15X)

CD-91-53911

Viscoplastic models are complex. The choice of an existing model, or the development of a new one, will depend on both the selected material and its intended application. One must know what the dominate material behaviors are for the selected material. Then one must determine which of these dominate material characteristics will manifest themselves in the intended application and which ones will not. The simpler the model, the better it is. Once a model is chosen and the material is characterized, the model can be implemented into a finite element code, thereby allowing a structural analysis of the intended application to be made. An example of this procedure is given for an engine nozzle liner. In applications the nozzle liner ratchets, causing a thinning of the liner and its eventual rupture. A viscoplastic analysis of the nozzle is in qualitative agreement with this observation.

Numerical Integration Development

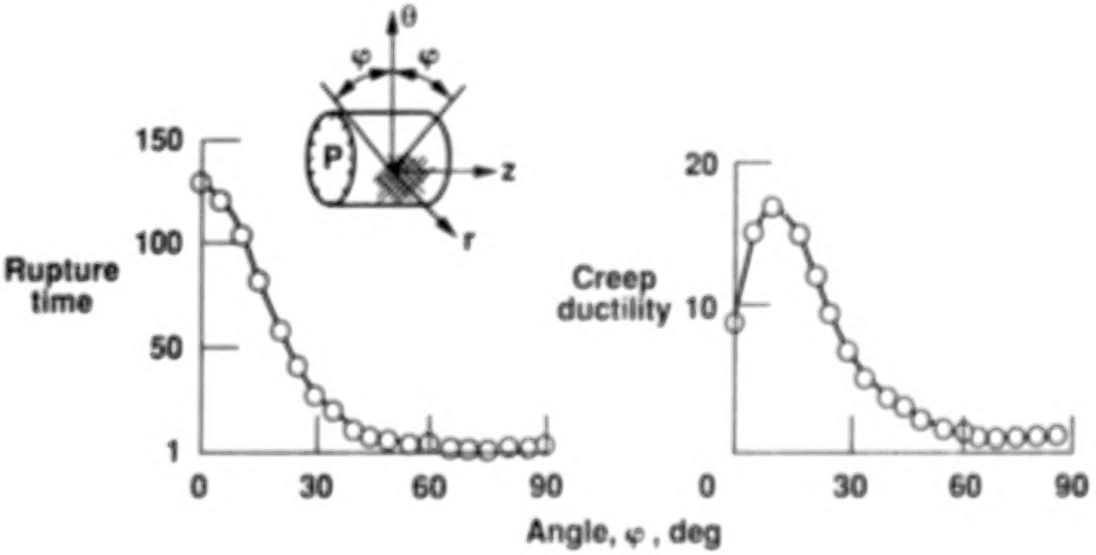


CD-91-53910

High-temperature fatigue data for metallic and intermetallic matrix composites, MMC's and IMC's, are very scarce. In the search for a correlating parameter for fatigue, Bartolotta (of NASA Lewis) recently proposed a maximum strain model for fibrous composites. The data presented here are for SiC/Ti-24Al-11Nb at 425 and 815 °C. There appears to be no dependence on the mode of loading when this correlating parameter is used. Region I is defined by the tensile ductility range and represents the domain where composite strength is dictated by fiber strength. Region II defines a progressive fatigue domain, where a decrease in the maximum strain leads to an increase in the fatigue life. This region is temperature dependent. Region III defines what appears to be a fatigue limit for the composite.

Creep Rupture

Damage Mechanics of Composites



CD 91-53914

Continuum damage mechanics was originally developed for modeling creep rupture and works well for that purpose. In work supported by NASA Lewis a continuum damage mechanics model has been developed for modeling creep rupture in fibrous composites that have elastic fibers and a viscoplastic matrix. The model was developed by using an isochronous damage function, which is assumed to depend on the invariants specifying the maximum tensile stress normal to a fiber and the maximum longitudinal shear stress along it. These stress components lead to damage in the composite at the fiber-matrix interface and eventually cause creep rupture. The theory has been applied to thin-wall pressure vessels having one and two families of helically oriented fibers. The results shown are for two fiber families. Results for one fiber family are similar, with the maximum creep ductility shifted out to a fiber angle of about 30°.

SUMMARY

- Metals at elevated temperature exhibit complex, inelastic, hereditary behavior.
- Over the past decade through the HOST, SSME, and HITEMP programs, NASA Lewis has advanced the state of the art in viscoplastic analysis and experimentation.
- The theoretical, computational, and experimental personnel at Lewis have extensive and unique experience in the following areas:
    - High-temperature experimentation
    - Thermomechanical fatigue
    - Life assessment
    - High-temperature structural analysis
    - Viscoplastic constitutive modeling
    - Composite mechanics

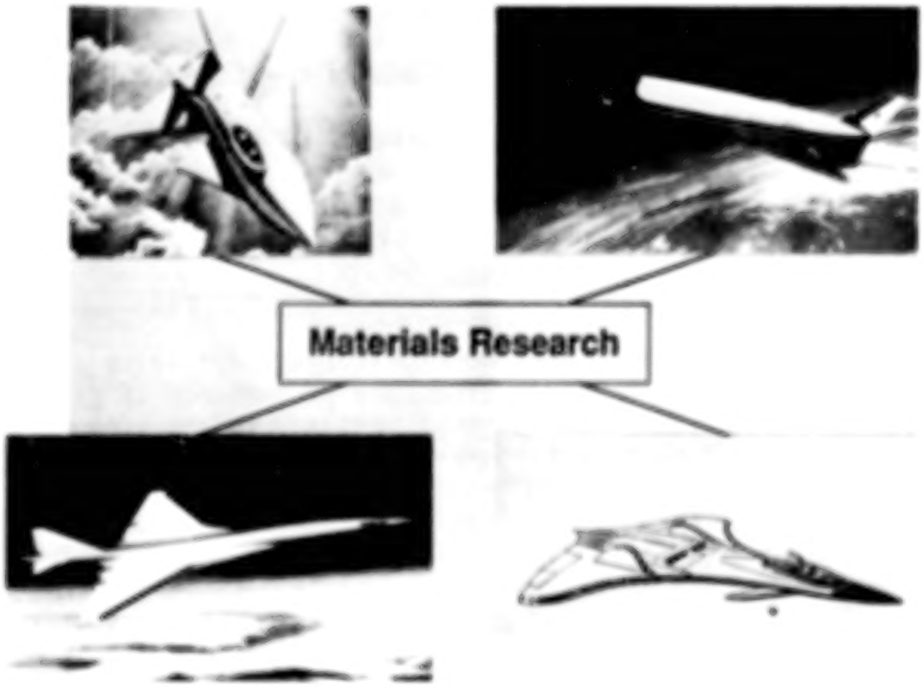
N91-20109

OVERVIEW OF LEWIS MATERIALS RESEARCH - CONTRIBUTIONS, CURRENT EFFORTS, AND FUTURE DIRECTIONS

Salvatore J. Grisaffe and Carl E. Lowell
NASA Lewis Research Center
Cleveland, Ohio

In the 1940's, materials research efforts on high-temperature valve alloys at NASA Lewis Research Center led to the improved lives and performance of piston-engined fighter and bomber aircraft. The metallurgical skills acquired from that work were then applied to the formulation of high-temperature turbine blade alloys for aircraft gas turbine engines developed in the 1950's. Today, the benefits of superior U.S. engine technology have never been clearer for both commercial and military aircraft. Yet, these superior engines of the 1980's and 1990's now use some of the materials that we helped conceive and evolve in the 1960's and 1970's. NASA Lewis is currently charged with helping industry create, advance, and develop the materials for 21st century engines - 2005 to 2015+. In this paper, we present highlights of our past work, our staff, and our facilities. We also summarize the challenges and new material and process concepts we are working on now as well as our vision for future efforts.

Advanced Systems Won't Fly Without Advanced Materials



Materials Research

CD-91-[illegible]

From advanced hypersonic aircraft to supersonic fighters and commercial transports, tomorrow's aircraft will stretch the limits of materials technology. Both airframe and engine materials and structures will require extended temperature capability in conjunction with the same or longer life compared with today's systems. Practical advanced engines require hot, highly loaded rotating and static components which must resist very aggressive oxidation-corrosion environments. Temperatures are rising, lives must be long, and weight must be reduced.

Thus we face many tough challenges as we pursue the advanced revolutionary technologies that offer the potential solutions to our problems: that is, high-temperature composites. Furthermore, instead of the 15 to 20 years that it has usually taken for materials to develop from concept to commercialization, we now are being asked to deliver these materials in only 10 to 12 years. Thus, the requirements are great, and only a very strong collaborative effort between the Government, industry, and academia will let us reach our goals.

We're a Major Government Aerospace Materials Laboratory

- Continued output of technology that is used by industry
  - PMR polymer composites
  - Thermal barrier coatings
  - MCrAl's
  - Ceramic composites
  - Solid, high-temperature lubes
  - Metal matrix composites
- High-quality staff
- Strong in-house capability
- Solid, collaborative ties to industry and key universities

CD-91-54059

Lewis Research Center's Materials Division is NASA's major high-temperature materials research group. It has a long history of technical contributions that have been used by the aerospace industry. Such contributions are the result of considerable efforts to recruit and retain top researchers, to provide facilities and laboratories that effectively support them, and to interact well with industry and universities to facilitate the exchange of new developments.

Our Job

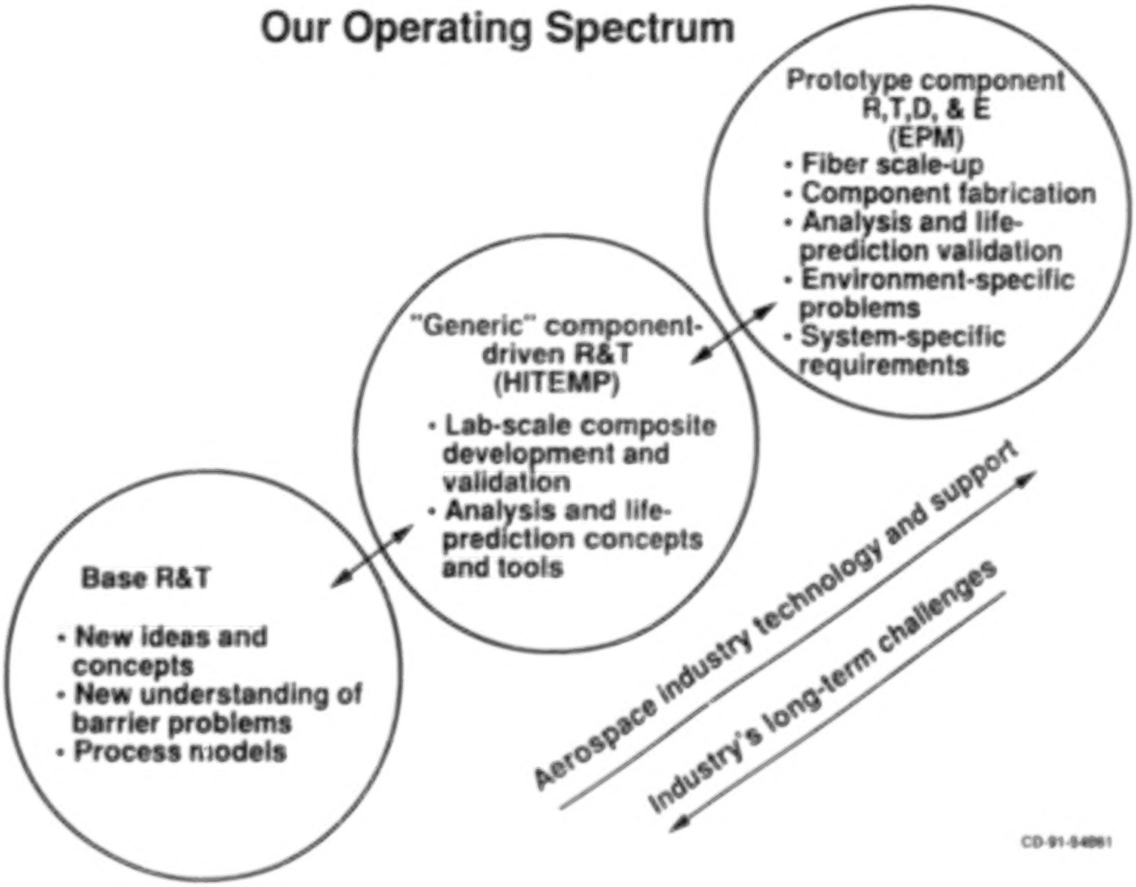
- New materials, insights into technical challenges, and process models for the next generation of NASA and aerospace industry needs
- Technology acceleration via cooperative, collaborative research with industry
- "Generic component" materials and process technology options and rapid technology transfer to support conversion of ideas into U.S. competitive systems
- Direct support of major NASA flight projects

CD-91-54060

Our Technology is Used by Industry, and We Work on Barrier Problems

CD-91-54062

Our Operating Spectrum



Prototype component
R,T,D, & E
(EPM)
- Fiber scale-up
- Component fabrication
- Analysis and life-prediction validation
- Environment-specific problems
- System-specific requirements

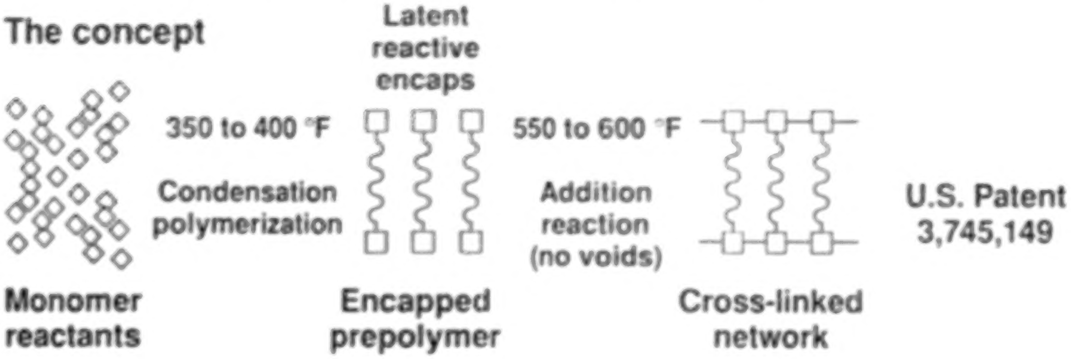
"Generic" component-driven R&T
(HITEMP)
- Lab-scale composite development and validation
- Analysis and life-prediction concepts and tools

Base R&T
- New ideas and concepts
- New understanding of barrier problems
- Process models

Aerospace industry technology and support
Industry's long-term challenges
CD-91-54061

New ideas generated by our base research and technology which meet generic component needs receive focused development and evaluation efforts. Promising materials systems are further optimized, scaled-up, and transitioned into prototype components for detailed industry assessment. From such work, new long-term barrier problems are identified and act as a guide for new base research and technology efforts.

PMR Family of PMC's for 500 to 700$^+$ °F



The concept

Monomer reactants

350 to 400 °F
Condensation polymerization

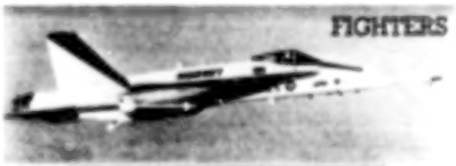
Latent reactive encaps

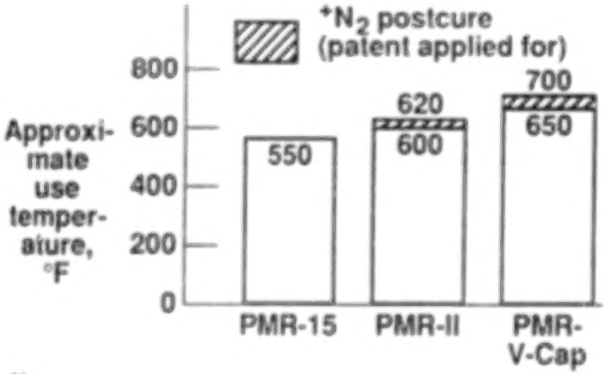
Encapped prepolymer

550 to 600 °F
Addition reaction (no voids)

Cross-linked network

U.S. Patent 3,745,149





- Prepregs available commercially

CD-91-54063

In the late 1960's and early 1970's, Lewis researchers developed a new concept, a two-step polymerization of monomeric reactants (PMR) process to make polyimide composites. This process allowed liquid and gas reaction products to escape the composite structure prior to final cross-linking. As a result, fewer voids were retained, and processing became more flexible for high-temperature polyimide resins. Such resins are usually intractable. Through collaboration with the U.S. Navy and General Electric Company, PMR-15 engine ducts are flying in the F404 engines that power the Navy's F-18 Hornet fighters. These ducts save about 30 percent of the total weight and cost of the previously used, chemically-milled titanium ducts. Recent advances by Lewis scientists have increased polymer molecular weight, have added more thermally stable end-caps, and have, by a nitrogen post cure, substantially raised the glass transition temperature. These advances, in turn, allow higher ultimate use temperatures. Prepregs of PMR-15, PMR-II (increased molecular weight), and PMR-VCP (vinyl end-cap) are now available commercially.

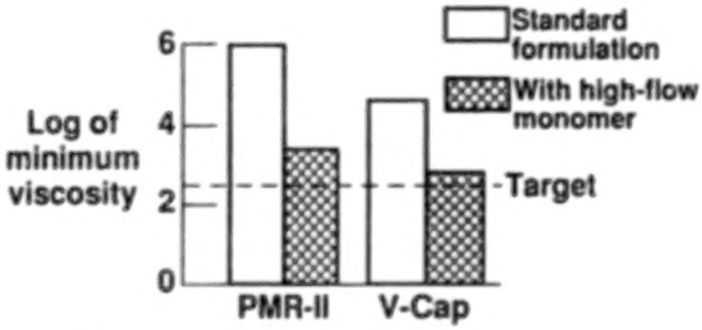
Polymer Composites - Current Efforts for Future Options

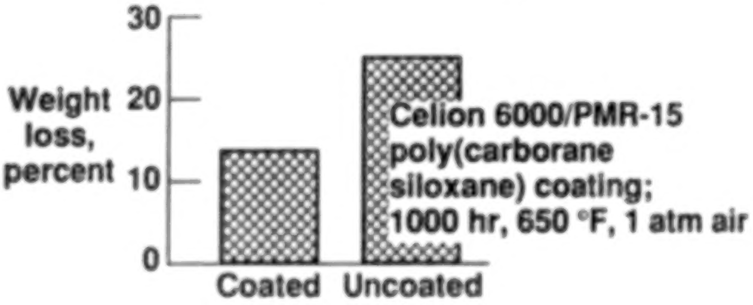
Challenges

- Higher flow resins to ↑ fabricability (and ↓ cost)
- Chemistry and surface control to ↑ oxidation resistance

Progress

Improved resin flow and processability



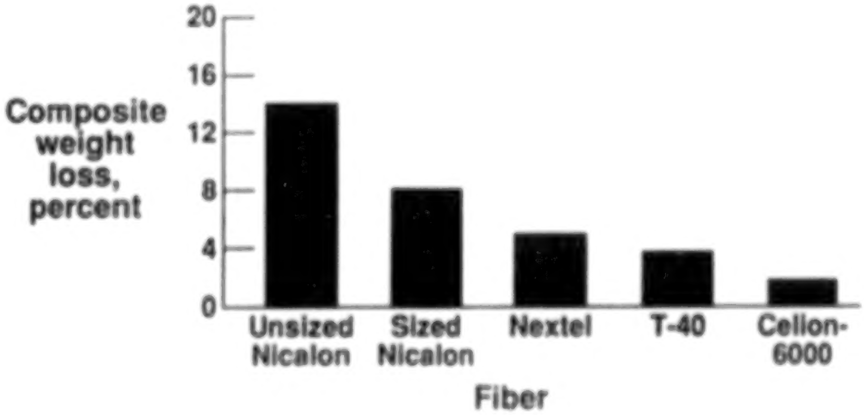


Celion 6000/PMR-15 poly(carborane siloxane) coating; 1000 hr, 650 °F, 1 atm air

CD-91-54064

- Fiber/matrix interface control to ↑ long-term properties

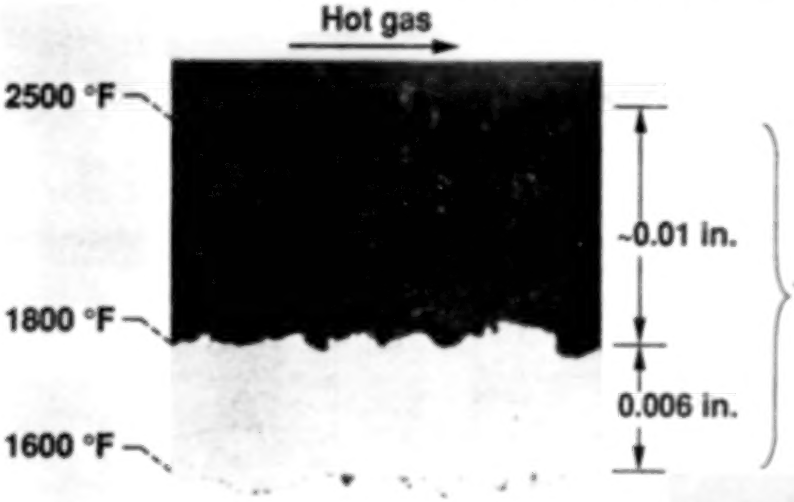
Composite weight loss



CD-91-54065

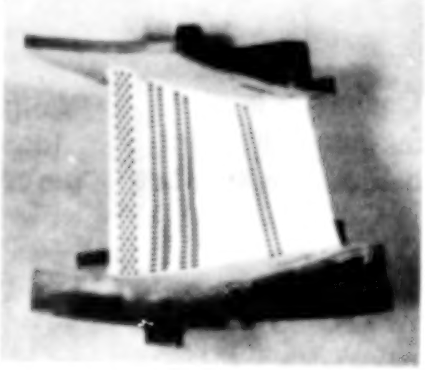
The major challenges we are facing include cost reduction and extended oxidation and environmental resistance. To reduce fabrication costs, we are using high-flow monomers to lower resin viscosity and make it easier to produce polymer matrix composites (PMC's) at lower temperatures or reduced autoclave pressures. To improve oxidation resistance, we are exploring a number of organic and inorganic surface coatings as well as trying to understand and control fiber/matrix reactions to preclude rapid oxidation attack along these interfaces.

Thermal Barrier Coatings (TBC)



U.S. Patents

4,055,705
4,535,033
4,255,495



TBC - coated vanes
in commercial
transport service

CD-91-[illegible]

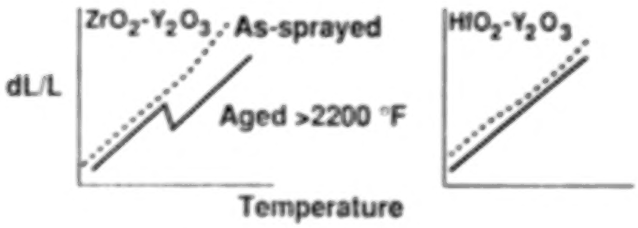
Thermal barrier coatings offer a way to operate air-cooled superalloy vanes, blades, and other aircraft components in high heat flux environments while maintaining reasonable substrate temperatures. The early Lewis Research Center work identified the plasma-sprayed two-layer coating (ZrO_2-$6Y_2O_3$ to ZrO_2-$8Y_2O_3$) partially stabilized zirconia outer coat deposited on a NiCrAlY oxidation-resistant bond coat layer. This coating has been embraced by industry, and components protected by variations on it are flying in many aircraft engines.

Thermal Barrier Coatings- Current Efforts for Future Options

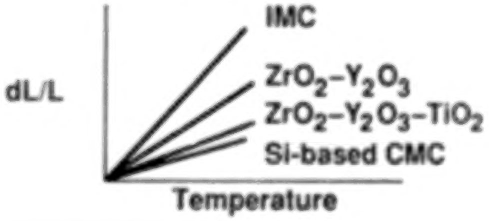
Challenges

Progress

- Resistance to higher surface and bond coat temperatures



- Useful life on either high or low C.T.E. substrates



- Reliable performance

Process control for
- Reproducibility
- Productivity
- Quality
- Safety

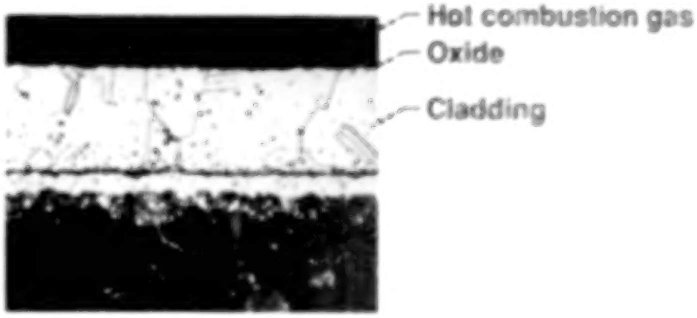
Allows
- Meaningful optimization

CD-91-54067

The challenges for more extensive thermal barrier coatings (TBC) use include higher temperature resistance, ability to resist spalling, and general reproducibility of performance. We have been studying stabilized HfO_2 systems which appear to be more resistant to eventual destabilization of the oxide and the resultant expansion inversion that occurs during temperature cycling.

Further alloying of the oxides provides one way to modify CTE's for greater spall resistance. To develop more reproducible systems, we are currently evaluating robot-controlled, rather than hand-held, plasma spray systems.

MCrAl's as Protection Systems



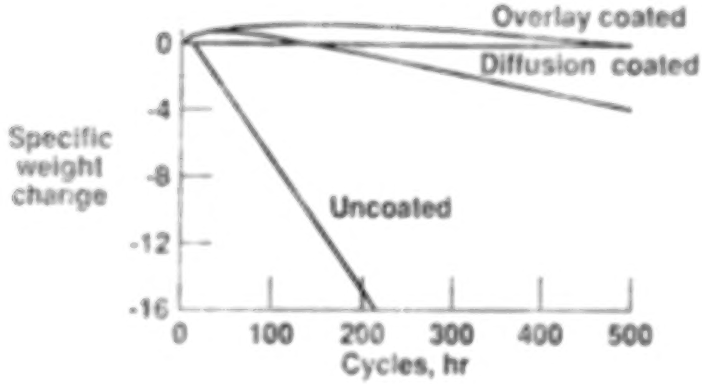
U.S. Patents

4,451,496
4,446,199
3,869,779

Cyclic oxidation mechanisms

Cladding (BMI) and testing

Most engines



CD-81-54066

We previously discussed the two-layer TBC system. The success of such coatings can be, in part, traced to earlier Lewis work on cyclic oxidation fundamentals of a variety of nickel and iron alloys. We found a clear and overwhelming benefit of reasonably high combined Cr and Al levels in such alloys on their ability to form thin, adherent alumina scales which rapidly reformed if they spalled. This led to the study of Cr/Al balance; the study of cycle temperature, hold time, cooling time, and other effects; and attempts to apply promising alloys as clads via HIP processing (the latter work was done contractually for Lewis by Battelle Memorial Institute). The performance of such clad superalloys - their long-term cyclic oxidation protection - helped motivate the industry to use current physical vapor deposition (PVD) overlay coatings for airfoils.

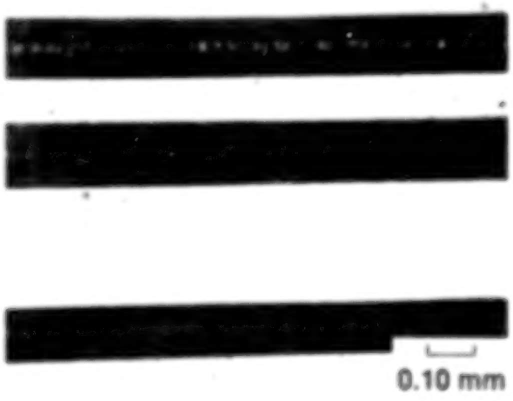
OF POOR QUALITY

NiCrAl's - Current
Efforts for Future Options

Challenges

- Reinforce weak, but oxidation resistant, material for structural use

Progress



Al_2O_3/FeCrAlY

CD-91-54069

As an alternative option, we are attempting to eliminate the substrate entirely and reinforce the weak, but highly oxidation resistant, former coating and cladding alloys. This concept offers some interesting potential, and early tests show good resistance to thermal cycling.

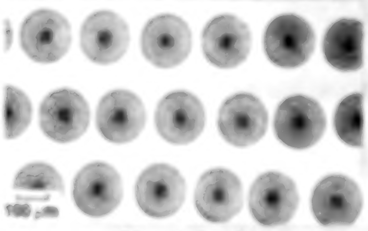
ORIGINAL PAGE IS
OF POOR QUALITY

Composite Fabrication

Arc spray

No commercial applications yet

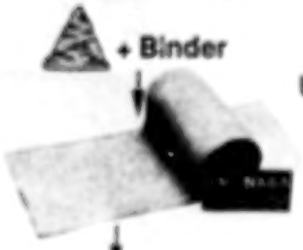
SiC/Ti$_3$Al+Nb composites fabricated from arc-sprayed monotape



Unetched

SiC/RBSN

Powder cloth



U.S. Patent 4,689,188

CD-91-54070

Numerous advanced fabrication processes have been developed in the Materials Division. The Lewis-patented arc spray method and the "powder cloth" approach have been used to make intermetallic and ceramic monotapes. Such tapes can then be angle plied, laid up, and hot pressed or HIPed to the final desired density and near-net shape.

R QUALITY

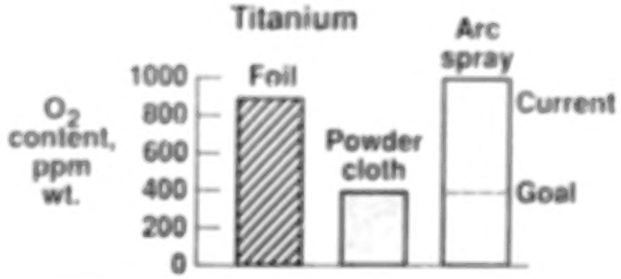
Composite Fabrication-
Current Efforts for Future Options

Challenges

- Minimal process contamination

Progress

Titanium



- Complex shapes to near-net shape

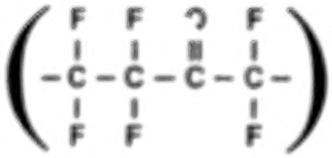
CD-91-54071

Monotapes offer a low-cost approach to composite fabrication, and arc-sprayed monotapes offer an even lower total cost. Our efforts to minimize material contamination during monotape fabrication have been progressing well. Titanium composites have been made with less than half the oxygen content of composites made by the foil-rolling approach. The lower cost, arc-sprayed titanium composites are equal in oxygen to those made by the foil process, but we have ideas that we think will let us reach an oxygen level near that of powder-cloth composites.

Fabrication to near-net shape is critical in minimizing composite component costs. Early efforts to fabricate metal matrix composite blades (NASA contract with TRW) and tubing (Lewis) provided us new insights into such processes.

ORIGINAL PAGE IS
OF POOR QUALITY

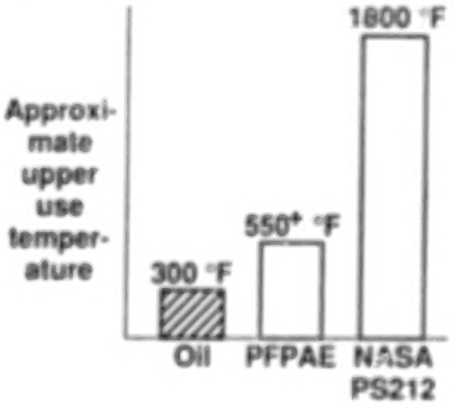
Trying to Meet Severe Lubrication Requirements



$$\left(\begin{array}{c}\mathrm{F}\\|\\-\mathrm{C}-\\|\\\mathrm{F}\end{array}\begin{array}{c}\mathrm{F}\\|\\\mathrm{C}-\\|\\\mathrm{F}\end{array}\begin{array}{c}\mathrm{O}\\\|\\\mathrm{C}-\\ \\ \end{array}\begin{array}{c}\mathrm{F}\\|\\\mathrm{C}-\\|\\\mathrm{F}\end{array}\right)$$

Perfluoro-polyalkyl-ethers

U.S. Patent 4,728,448



CD-91-54072

Aircraft engines that require ever-higher operating temperatures also require liquid lubricants that function well under increased temperatures. Such higher temperature lubricants also minimize total engine weight because of their reduced need for auxiliary oil coolers. These lubricants may also better resist lubricant decomposition during service. Our work focuses on understanding the stability of candidate perfluoropolyalkylethers (PFPAE) and exploring ways to increase their stability. In addition, we are also looking for solid lubricants for advanced gas turbines, Stirling engines, and internal combustion engines that can operate at much higher temperatures. NASA has developed a solid lubricant family called PS/PM200 that has an operating potential to 1800+ °F.

High-Temperature Lubricants-
Current Efforts for Future Options

Challenges

- High-temperature strength, oxidation, and reactions

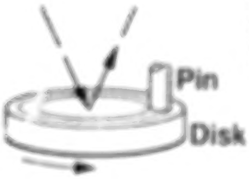
Progress

PM-200

Strength at room temperature ≈ bronze

Strength at high temperature ≈ stainless steel

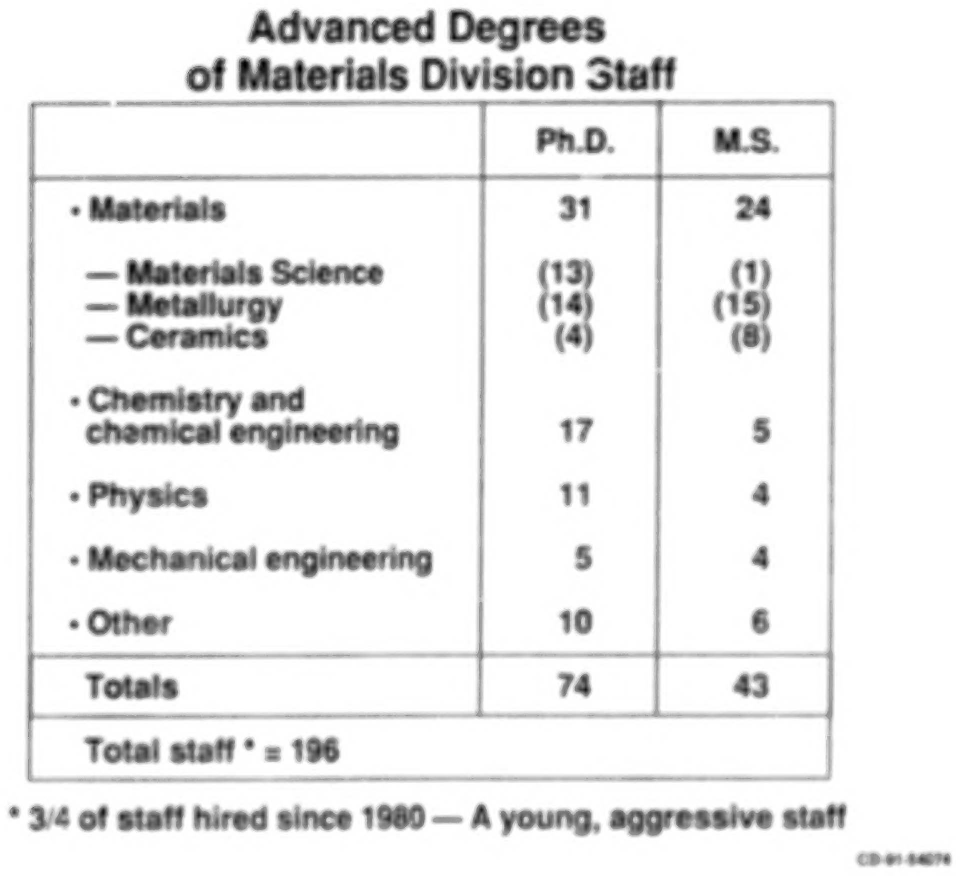
ESCA



Surface oxide catalyzed PFPAE lubricant decomposition

CD-91-54073

Since high-temperature lubricant properties such as strength (for solid lubes), oxidation resistance, and reactivity are barriers to even hotter bearings, we are focusing on these problems. Currently, a powder metallurgy (PM) version of our plasma-sprayed (PS) solid lubricant is showing promise with strengths equal to many of the current bearing materials. Our work on liquid lubricants is currently focused on basic understanding of reaction mechanisms, as well as on minimizing the catalytic decomposition of PFPAE caused by surface films on metals.



Advanced Degrees of Materials Division Staff

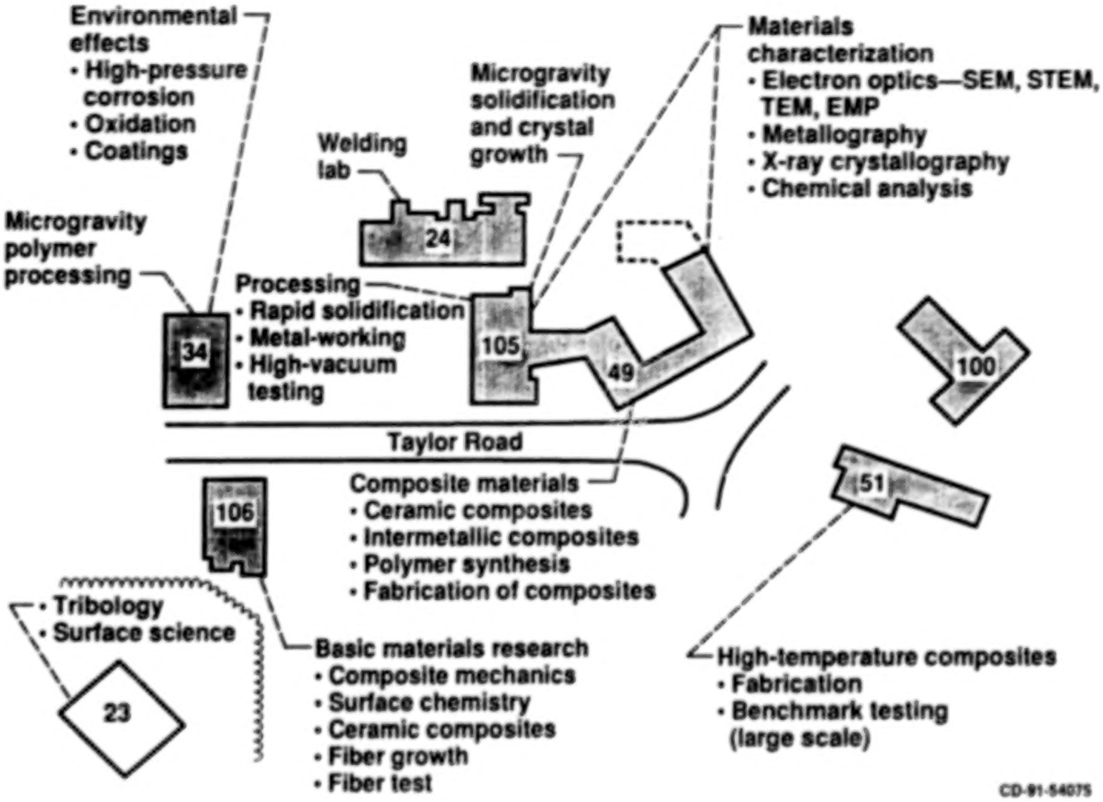
	Ph.D.	M.S.
• Materials	31	24
— Materials Science — Metallurgy — Ceramics	(13) (14) (4)	(1) (15) (8)
• Chemistry and chemical engineering	17	5
• Physics	11	4
• Mechanical engineering	5	4
• Other	10	6
Totals	**74**	**43**
Total staff * = 196		

* 3/4 of staff hired since 1980 — A young, aggressive staff

CD-91-[illegible]

We have a high quality, young staff with a broad range of backgrounds and skills. Currently, of 196 total staff, 74 hold Ph.D.'s and 43 have M.S.'s. Although, as expected, our heaviest concentration of expertise is in the materials area, our staff is well balanced with top chemists, physicists, mechanical engineers, computer scientists, and aeronautical engineers. Note that approximately 75 percent of our staff has been hired within the last 10+ years, so our skills are very much up-to-date.

Lewis Materials Research Areas



Environmental effects
- High-pressure corrosion
- Oxidation
- Coatings

Microgravity polymer processing

Welding lab

Microgravity solidification and crystal growth

Materials characterization
- Electron optics—SEM, STEM, TEM, EMP
- Metallography
- X-ray crystallography
- Chemical analysis

Processing
- Rapid solidification
- Metal-working
- High-vacuum testing

Composite materials
- Ceramic composites
- Intermetallic composites
- Polymer synthesis
- Fabrication of composites

- Tribology
- Surface science

Basic materials research
- Composite mechanics
- Surface chemistry
- Ceramic composites
- Fiber growth
- Fiber test

High-temperature composites
- Fabrication
- Benchmark testing (large scale)

CD-91-54075

We have a large, comprehensive materials research complex. We can fabricate, melt, cast, extrude, roll, weld, machine, test, and characterize all the systems we are investigating. These facilities plus our excellent staff offer industry and academic scientists and engineers a wide range of opportunities for research collaboration on problems of mutual interest.

Our Vision of the Future

- Continuing major R&T contributions - ideas and understanding
  - PMC
  - IMC
  - MMC
  - CMC
  - Fibers and coatings
  - Lubricants

  Raise use-temperatures, extend life, lower fabrication cost

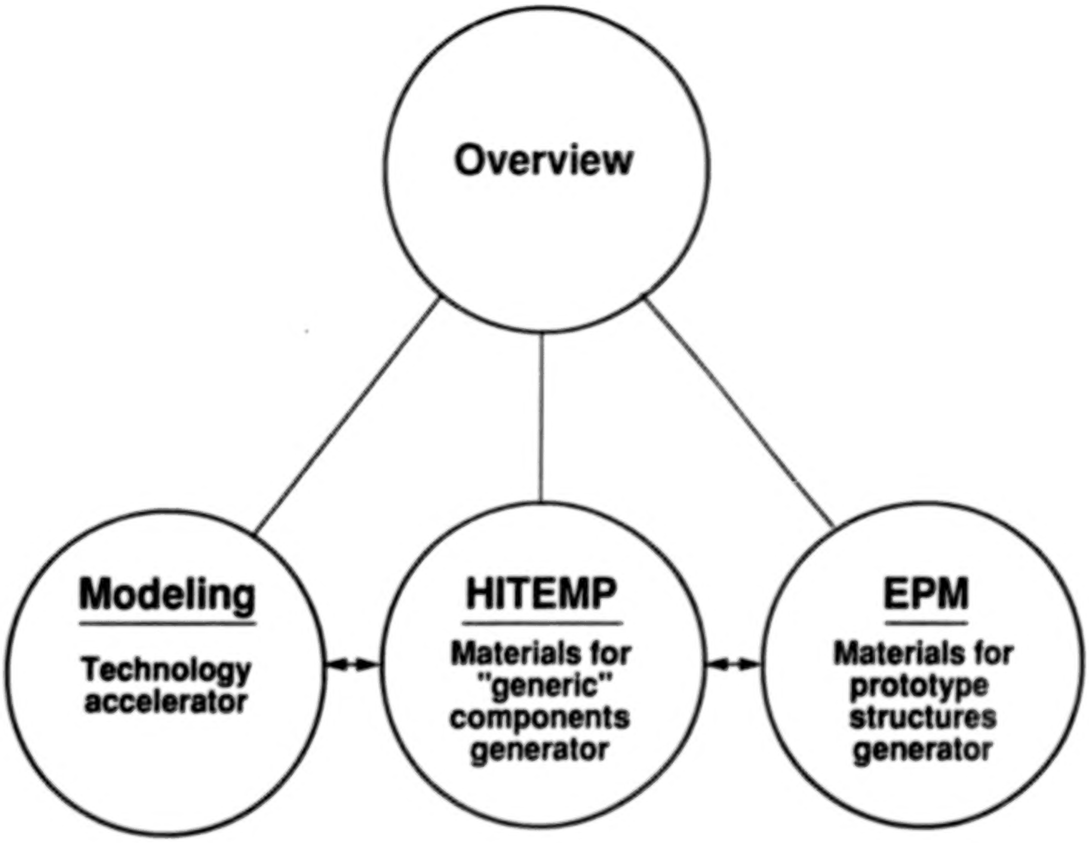
- More rapid movement of U.S. ideas to commercial and flight use
  - Process modeling to guide process optimization
  - Very close industrial collaboration on key opportunities – HITEMP, EPM, base R&T

- A growing national resource for high-temperature materials collaboration (call S. Grisaffe (216)-433-3193)

CD-91-54076

In the future, we will continue work to extend the use temperatures and operating lives of advanced composite engine materials while seeking new processes and process models with the potential to shorten time-to-process optimization and help lower costs.

We are dedicated to collaborating with industry, universities, and other agencies so that new ideas move rapidly into commercial and military systems. We are also dedicated to improving the competitive advantage of U.S. aerospace industries and the U.S. preeminence in world markets.



Overview

Modeling

Technology accelerator

HITEMP

Materials for "generic" components generator

EPM

Materials for prototype structures generator

CD-91-54077

We will now describe several specific projects aimed at generating and accelerating technology flow to U.S. industry. In our process modeling effort, we are working to assure a sound understanding of key interactive process variables and then to reduce the time to optimize processes. This is a key factor that can reduce the time from material and process concept to commercialization. Our other modeling efforts focus on microstructure versus property relationships in complex alloys and composites.

Our HITEMP (Advanced High-Temperature Engine Materials) program is aimed at rapidly verifying the promise of new materials, processes, and analytical methods at the laboratory specimen level. Its focus is on materials with potential for "generic" classes of components - with emphasis on key property ranges and performance characteristics (e.g., compressor blades, disks, combustors, cases, turbine vanes, turbine blades, and other components).

EPM (Enabling Propulsion Materials) is a very focused program aimed at providing and scaling up the critical, long-lead-time materials for a U.S. high-speed civil transport engine. These materials include ceramic matrix composites for low NO_x combustors and intermetallic matrix composites for lightweight exhaust nozzles.

CONCLUSIONS

1. We have a solid "track record" of working with industry to accelerate the introduction of new technology into engines.

2. We have a strong staff and a clear vision of future engine needs.

3. We have a strong commitment to collaborate with industry to help keep the United States preeminent in aircraft gas turbine engine technology.

N91-20110

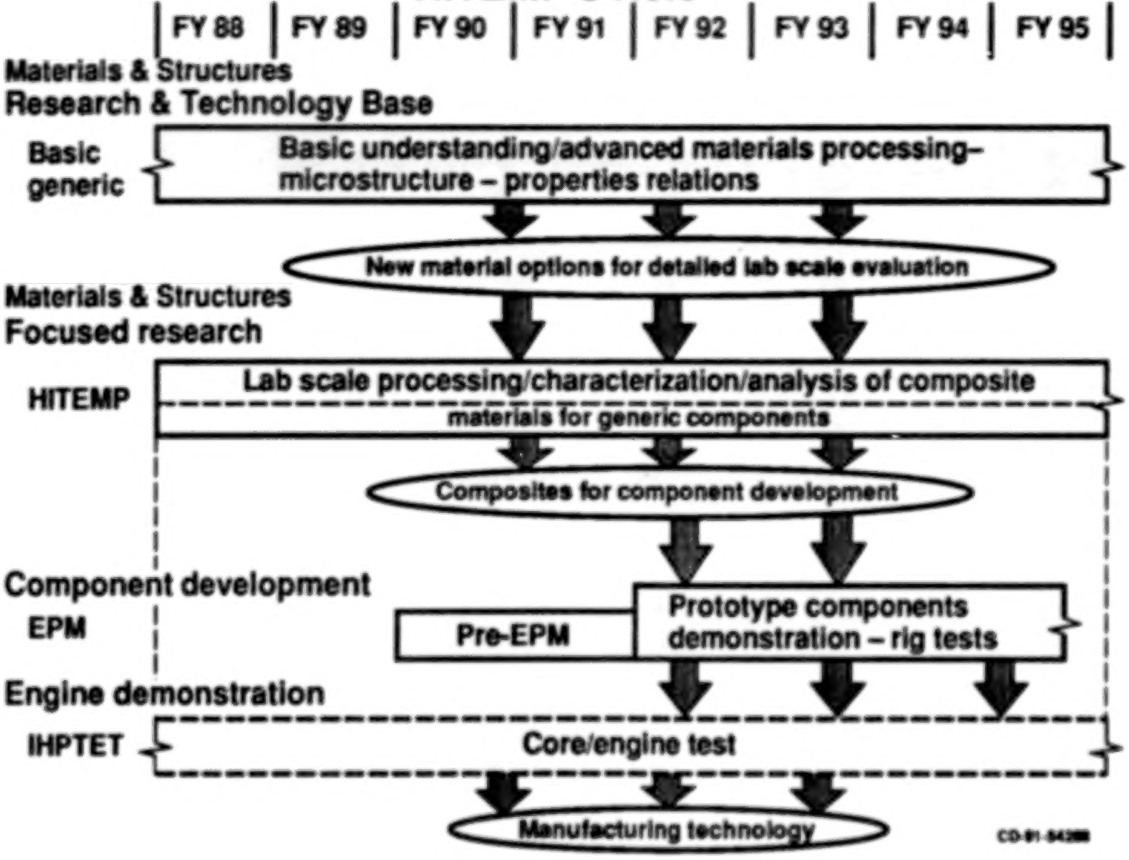
ADVANCED HIGH TEMPERATURE ENGINE MATERIALS TECHNOLOGY PROGRAM

Hugh R. Gray
NASA Lewis Research Center
Cleveland, Ohio

NASA's Advanced High Temperature Engine Materials Technology Program (HITEMP) is directed towards generating the technology for revolutionary advances in structural materials and analysis to enable the development of 21st century civil aeronautics propulsion systems. Major consideration is being given to propulsion systems that will be economical via reducing fuel consumption per passenger mile, reducing direct operating costs, extending life, and improving reliability. To achieve revolutionary advances in propulsion systems for 21st century transports, high temperature materials have been identified as the key technology to be addressed. The HITEMP Program is focusing on lightweight composite materials to gain revolutionary advances in the operating temperatures of advanced engines compared to the current state of the art. Emphasis is being placed on polymer matrix composites (PMC's) for potential use in fans, casings, and engine control systems. Intermetallic/metal matrix composites (IMC's/MMC's) are under investigation for application in such areas as compressor and turbine disks, blades, and vanes, and in the exhaust nozzle. For extremely high temperature applications, ceramic matrix composites (CMC's) are being explored. Initial applications may include liners for the combustor and exhaust nozzle, and turbine vanes and ultimately turbine blades and disks, or blisks.

One of the major distinctions between the HITEMP Program, which is focusing on civil transport aircraft propulsion systems, and programs such as the National Aerospace Plane (NASP) and IHPTET is the need for extended high temperature operation at maximum operating temperature. For civil aircraft engines, requirements are in terms of tens of thousands of hours, while for military applications they are in terms of thousands of hours and for NASP, which is a demonstration project, hundreds of hours will meet the goals. As an example of the long term requirement, civil applications, studies on the HSCT engine have indicated that 20 000 hr life with 90 percent at the maximum operating temperature will be required for economic viability. The long term life goal permeates the three classes of composite materials, PMC's, IMC's/MMC's, and CMC's, being investigated in HITEMP.

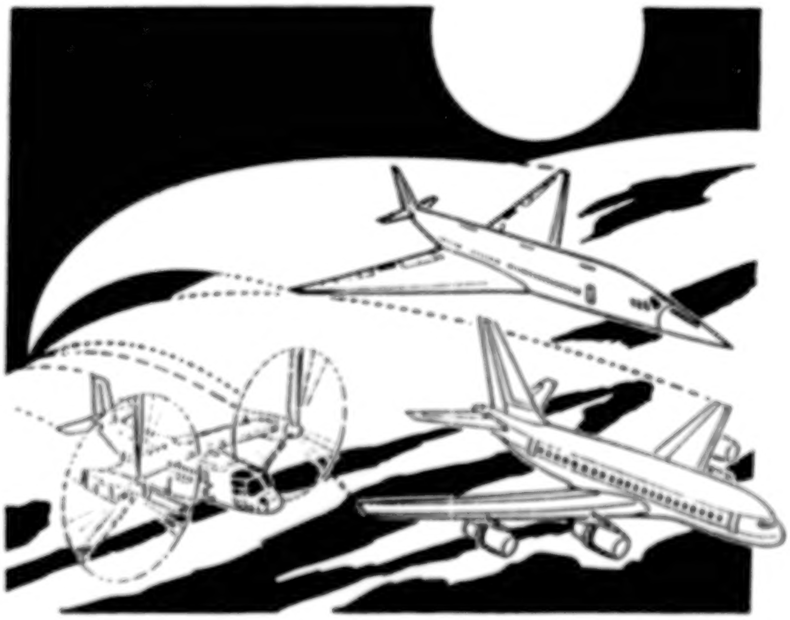
HITEMP's Role


FY 88
FY 89
FY 90
FY 91
FY 92
FY 93
FY 94
FY 95
Materials & Structures
Research & Technology Base
Basic generic
Basic understanding/advanced materials processing–microstructure – properties relations
New material options for detailed lab scale evaluation
Materials & Structures
Focused research
HITEMP
Lab scale processing/characterization/analysis of composite materials for generic components
Composites for component development
Component development
EPM
Pre-EPM
Prototype components demonstration – rig tests
Engine demonstration
IHPTET
Core/engine test
Manufacturing technology

The objective of the HITEMP program is to generate technology for revolutionary advances in composite materials and structural analysis to enable the development of 21st century civil propulsion systems with greatly increased fuel economy, improved reliability, extended life, and reduced operating costs. NASA considers this program to be a focused research effort that builds upon our basic research programs and that will feed results into application oriented projects such as the proposed NASA new initiative to develop the technology for a 21st century High Speed Civil Transport (HSCT). The Enabling Propulsion Materials (EPM) program is a major effort in the HSCT program and will utilize materials and structures concepts developed in HITEMP as well as elsewhere to provide the gains in engine materials that are required for economic viability and environmental acceptability. Also, HITEMP is closely coordinated with the joint DOD/NASA Integrated High Performance Turbine Engine Technology Program (IHPTET), and new materials from HITEMP may be utilized in future military applications.

Focus

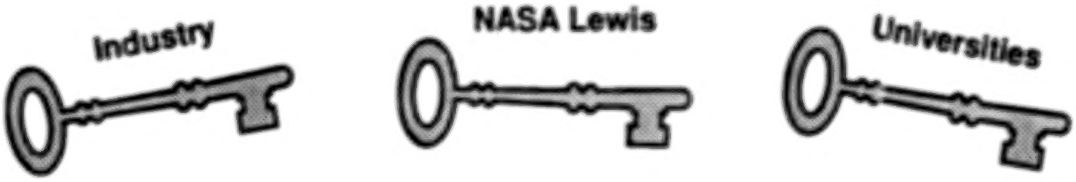
Engines for 21st century transport aircraft



CD-91-[illegible]

To help guide the research, we are funding preliminary design studies with Pratt & Whitney Aircraft, GE Aircraft Engine Co., and Allison Gas Turbine Division of General Motors. These three companies are studying high speed civil transport engines, ultra-high bypass ratio engines, and rotorcraft engines, respectively.

Cooperation – Key to Success



- Materials & property requirements for future engine components
- Design codes and validation needed
- Fiber development
- Composite fabrication
- Composite evaluation

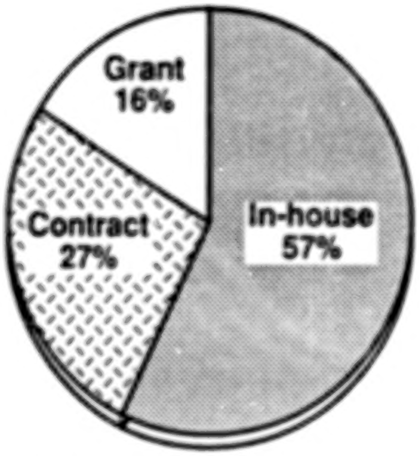
- Project management
- Fibers/matrices/composite options:
  polymers,
  intermetallics,
  and ceramics
- Analytical modeling
- Test methods

- Materials science
- Property characterization
- Structural modeling

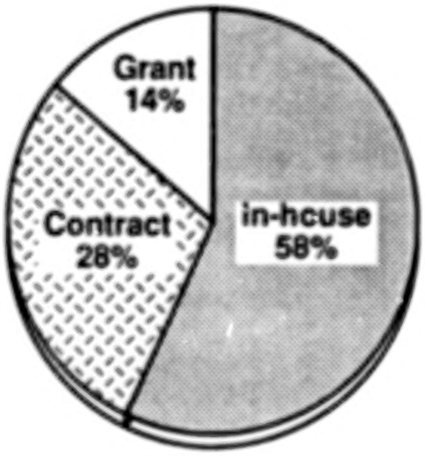
CD-91-[illegible]

The intent of HITEMP is to combine the skills of NASA Lewis researchers with those from industry and academia to achieve the objective of the program.

Funding Distribution



FY 90 Net R&D funds - $7018K
Total funds - $9216



FY 91 Net R&D funds - $7176K
Total funds - $9776

CD-91-54291

Based on FY'90 commitments and projected spending for FY'91, we estimate the distribution of spendable dollars over these 2 years to be about 28 percent in contracts, 15 percent as university grants, and the remainder in-house. A total of 14 contracts have been awarded via the NASA Research Announcement (NRA) route. Two of these have been renewed for a second year. In addition, two other contracts have been awarded by the competitive route, making a total of 16 HITEMP active contracts. An additional five contracts are currently being negotiated. The grant effort within HITEMP is distributed among 12 colleges and universities with some of them having multiple grant programs, for a total of 18 grants.

High Temperature Polymer Matrix Composites

Key issues

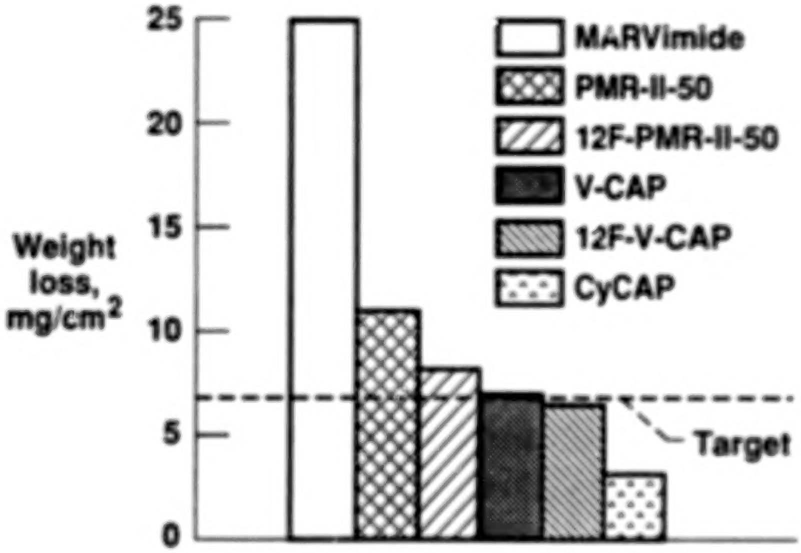
- Thermal-oxidative stability
  - Low weight loss
  - Good retention of mechanical properties
- Glass transition temperature
  - Must be at least 25 °C (50 °F) higher than intended upper use temperature
- Processability
- Resin-fiber interactions (interface)
  - Effect on thermal-oxidative stability
  - Effect on mechanical properties
- Oxidation-resistant coatings
- Accelerated aging

CD-91-54231

Polymer matrix composites are being developed to achieve a maximum operating temperature of 425 °C (800 °F). Matrices of interest include variations of PMR-II, MARVimide, V-CAP, and CyCAP. Key issues to be addressed in the future are oxidation resistant coatings and the need for conducting accelerated aging tests to determine the time-temperature operating envelope for the PMC's.

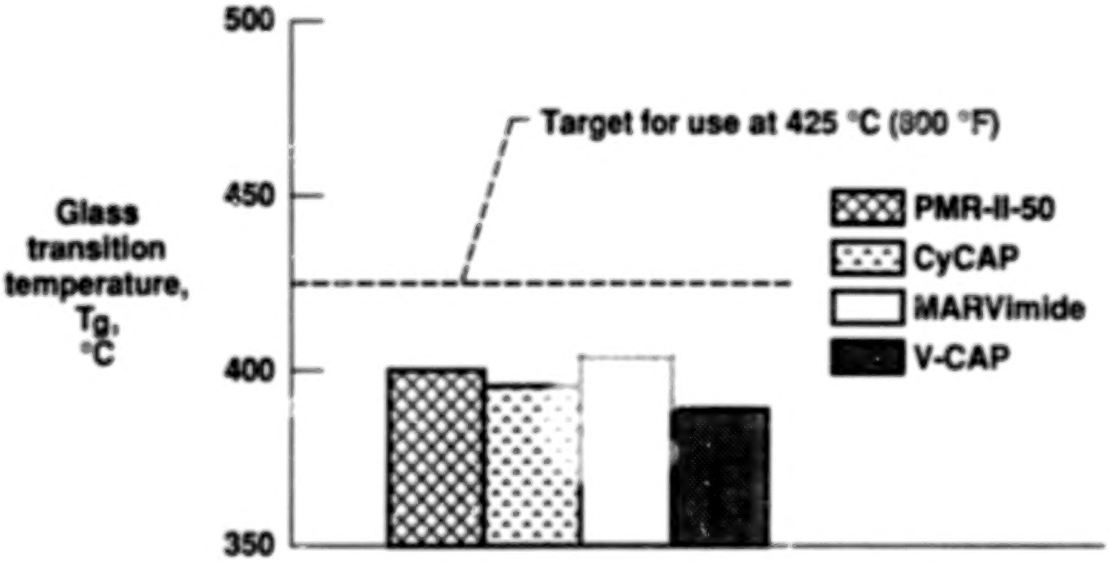
Comparison of Neat Resin Weight Losses for Addition Curing Resins

After 500 hr at 345 °C (650 °F), 1 atm Air



One of the key issues is to achieve thermal oxidative stability, i.e., low weight loss and retention of mechanical properties during the life of a specific component. Short time (500 hr) tests at 345 °C (650 °F) indicate that the CyCAP resin holds the most promise of the resins under investigation.

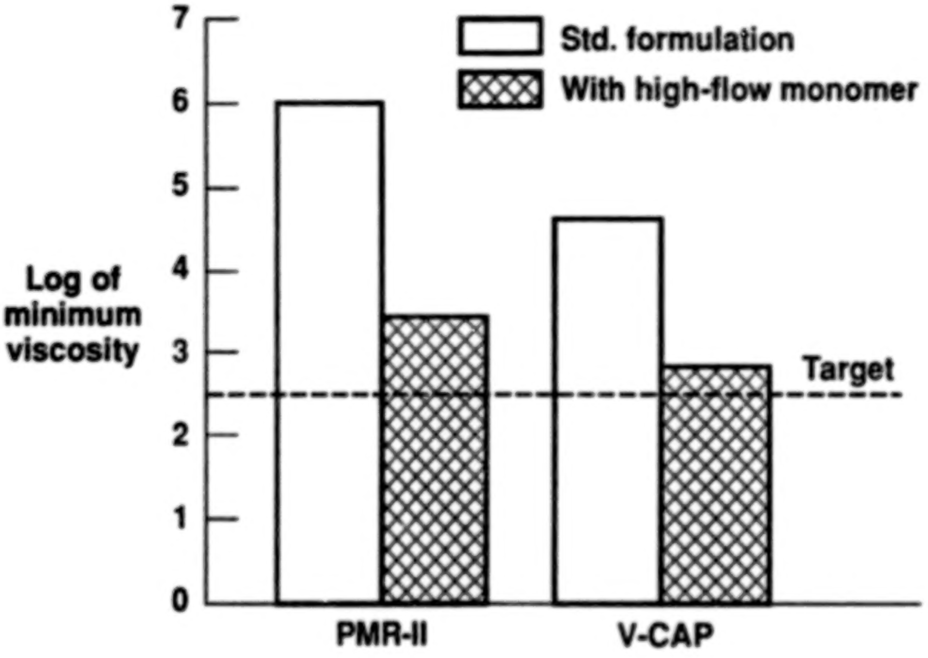
A Comparison of Glass Transition Temperatures of a Variety of Addition Polyimides



CD-91-54294

A second critical issue is the processability of the polymer resins. A glass transition temperature of 25 °C (50 °F) above the anticipated use temperature is desirable to maintain mechanical properties. Currently, of the resins studied, there does not appear to be a large variation with composition. On average, the transition temperatures are about 50 °C short of the goal.

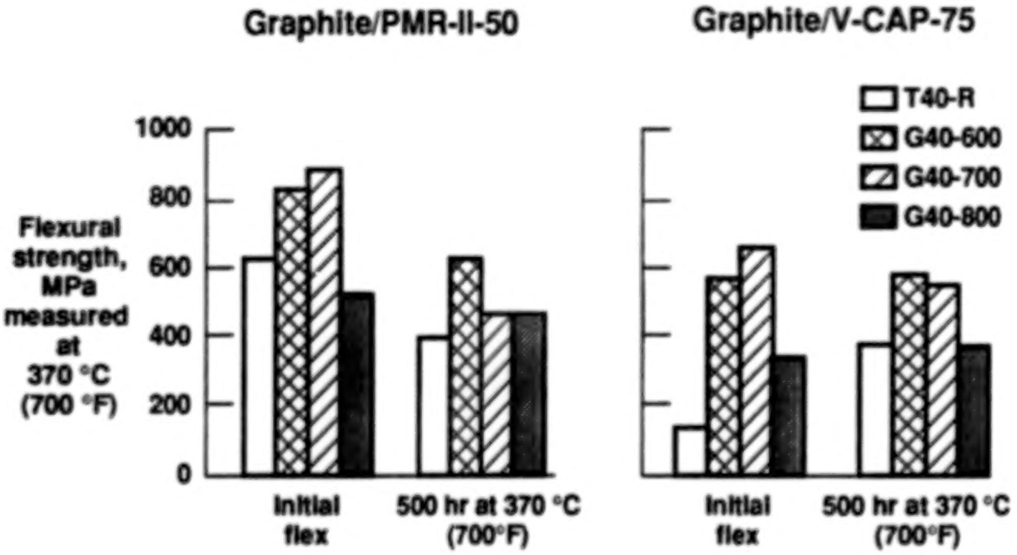
Use of High–Flow Monomer Enhances Resin Flow and Improves Processability



CD-81-54295

A second factor to be considered in processability is the resin's capability to flow as determined by viscosity measurements. A high flow monomer has been added to PMR-II and to V-CAP with successful results. In both cases a substantial reduction in viscosity was produced which will allow for more complete removal of volatile byproducts formed during curing. A final key issue under investigation is that of fiber-resin interaction during service. Depending upon the interface reaction, both thermal-oxidative stability and mechanical properties may be degraded.

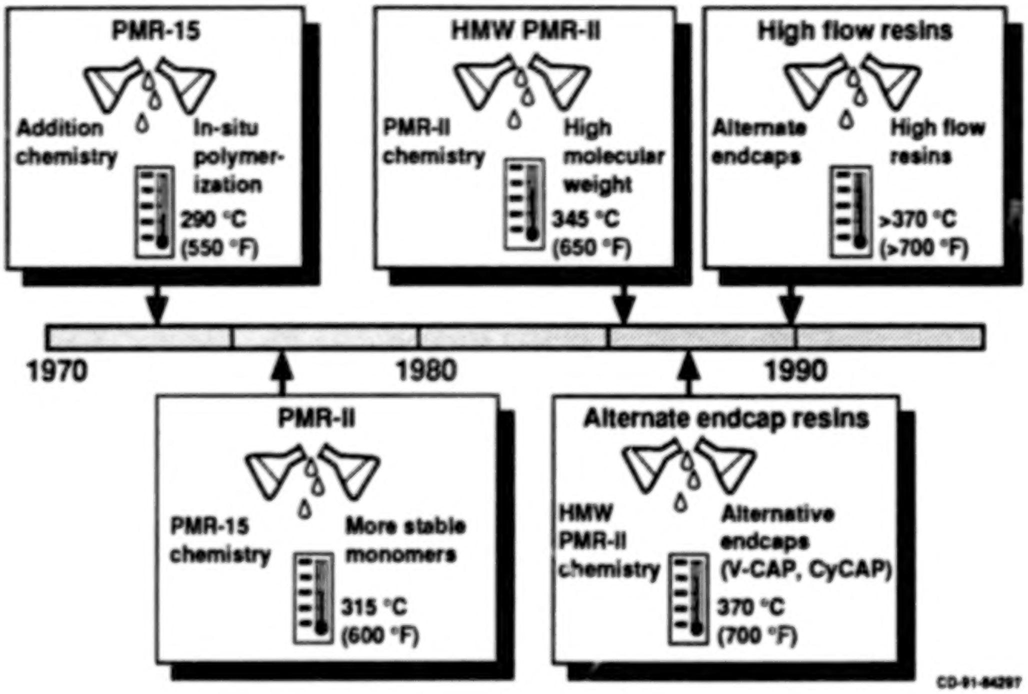
Choice of Graphite Fiber Greatly Influences the High Temperature Performance of PMC's



CD-91-54296

Results have shown that choice of graphite fiber for a specific matrix composition can greatly impact the strength of PMC's after high temperature exposure. PMR-II-50 and V-CAP-75 with the G40-600 fiber exhibited the highest flexural strength after exposure at 370 °C (700 °F) compared to the use of T40-R, G40-700, and G40-800 fibers.

Two Decades of High Temperature Polymer Research at NASA Lewis



PMR-15

Addition chemistry

In-situ polymerization

290 °C (550 °F)

HMW PMR-II

PMR-II chemistry

High molecular weight

345 °C (650 °F)

High flow resins

Alternate endcaps

High flow resins

>370 °C (>700 °F)

1970

1980

1990

PMR-II

PMR-15 chemistry

More stable monomers

315 °C (600 °F)

Alternate endcap resins

HMW PMR-II chemistry

Alternative endcaps (V-CAP, CyCAP)

370 °C (700 °F)

CD-91-54297

From the introduction of PMR-15 in the early 1970's to the present, PMC research at NASA Lewis has demonstrated an increase in use temperature from 290 °C (550 °F) to 370 °C (700 °F) for the advanced composites such as the high molecular weight PMR's and alternate endcaps such as V-CAP and CyCAP.

High Temperature Polymer Matrix Composites

Future research directions

- Use of high flow systems to maximize prepolymer molecular weight
- Use of more stable endcaps to improve thermal-oxidative stability
- Nitrogen postcuring to get high T_g's necessary for 425 °C (800 °F) goal
- Potential use of high temperature resins as fiber sizings

CD-91-54298

The use of high flow systems to maximize prepolymer molecular weight is viewed as the next fruitful area of research where use temperature may be further increased. Other future areas of research will emphasize nitrogen postcuring to achieve the desired glass transition temperature and along with coatings, in turn, achieve long term durability at an operating temperature of 425 °C (800 °F).

Intermetallic Matrix Composites

Key issues

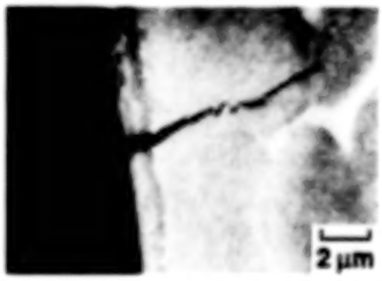
- **CTE mismatch**
- **Oxidation resistance**
- **Composite modeling/architecture**

CD-91-54299

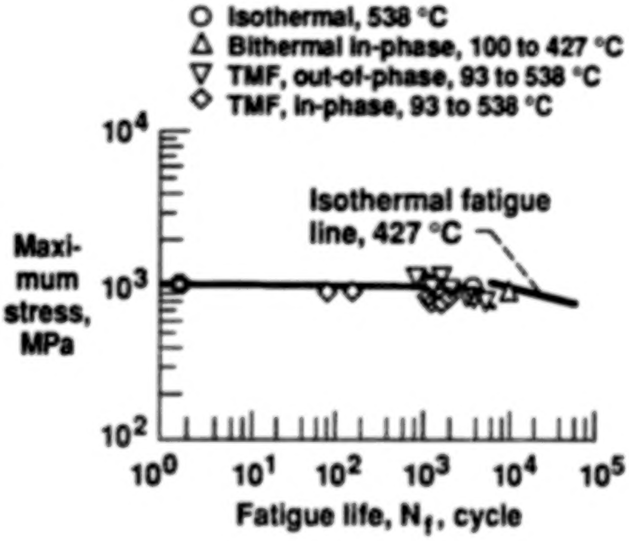
Metal/intermetallic matrix composites under investigation in HITEMP are slated for use temperatures that bridge the gap between PMC's and CMC's, or above 425 °C (800 °F) to about 1315 °C (2400 °F). Intermetallic compounds, especially the aluminides, provide a low density matrix and the potential for good oxidation resistance at the anticipated use temperatures. However, the aluminides have coefficients of thermal expansion (CTE's), ranging from about $10x10^{-6}$/°C for Ti_3Al to $23x10^{-6}$/°C for FeAl. Since the primary fiber currently available, SiC SCS-6, has a relatively low coefficient of thermal expansion ($5x10^{-6}$)/°C), the issue of fiber-matrix CTE mismatch is the key issue for IMC's.

Effects of CTE Mismatch

Growth of radial cracks in SiC/Ti_3Al due to 1000 thermal cycles between 85 and 815 °C



Thermomechanical fatigue life results for $[0]_9$ SiC/Ti-15-3-3-3

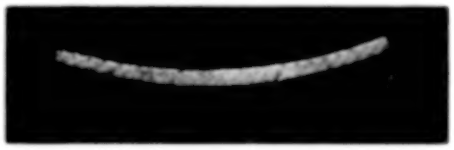
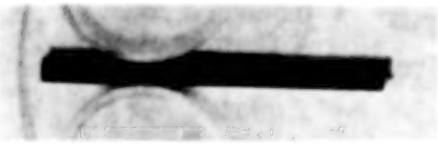


CD-91-54300

The detrimental effect of CTE mismatch is evidenced during both thermal cycling and thermomechanical fatigue (TMF) testing. For example, pre-existing radial cracks in SiC/Ti_3Al propagated into the matrix during thermal cycling in vacuum between 85° and 815 °C. In TMF testing cyclic lives were greatly reduced from those obtained in isothermal fatigue tests at both 427 and 538 °C. For in-phase TMF tests cyclic lives were reduced several orders of magnitude relative to out-of-phase TMF tests.

Experiments/Modeling to Understand CTE Mismatch

Thermally cycled FeAl and FeCrAlY composites

Composite system	23° to 825 °C, 1000 cycles (rate = 25 °C/s)	$\frac{\text{CTE matrix}}{\text{CTE fiber}}$, $10^6/°\text{C}$
FeAl/W		$\frac{23}{5} = 4.6$
FeCrAlY/W		$\frac{14}{5} = 2.8$
FeCrAlY/Al_2O_3		$\frac{14}{9} = 1.6$

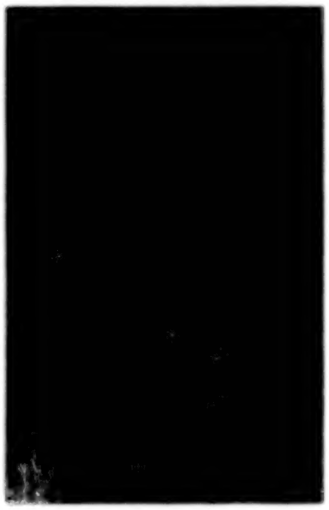
CD-91-84301

This problem manifests itself in thermal cycling where strength decreases and in thermal mechanical fatigue where life is degraded compared to isothermal tests. The severity of the problem is indicated by comparison of the degree of permanent deformation of composites with varying matrix-fiber CTE ratios. From these limited data, it appears that a ratio of 1.6 may be acceptable without severe degradation of the composite.

ORIGINAL PAGE IS
OF POOR QUALITY

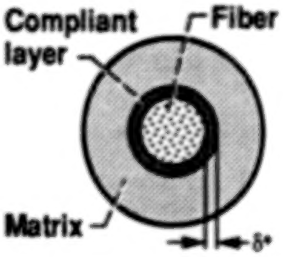
Concepts to Resolve CTE Mismatch

High CTE fibers



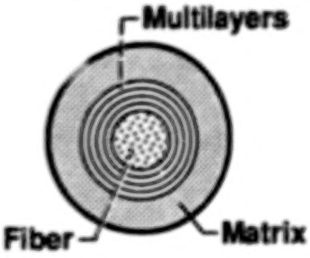
Sputtered Nb_2Be_{17} fiber

Compliant layer



- Joint effort with Textron Specialty Materials
- In-house & UCSB analytical modeling (concentric cylinder model, F.E.M.)
  - Thermal elastic
  - Thermoelastic-plastic
  - Shakedown

Graded or multilayer



- Joint effort with Textron Lycoming-Sarnoff
- Multilayer analysis, concentric cylinder model to be studied

CD-91-54302

A threefold approach is underway to overcome the CTE issue in IMC's, including identification of high CTE fibers, application of a compliant layer, and the use of a graded interface of multiple layers at the interface.

ORIGINAL PAGE IS
OF POOR QUALITY

Oxidation Resistance

Ti_3Al + Nb

- Pack aluminide ($TiAl_3$) coatings provide excellent protection on matrix material, but fail on composites.
- Ductile plasma sprayed NiCrAlY coatings are being studied.

FeAl

- Oxidation resistance is adequate for intended use temperature of 1000 °C.
- Zr-, B-, and Hf-doped alloys are being characterized to 1100 °C .

NiAl

- Zr-doped NiAl alloys must have more than 40 at.% Al for good oxidation properties to 1200 °C.
- Diffusion-based life prediction model is being developed.

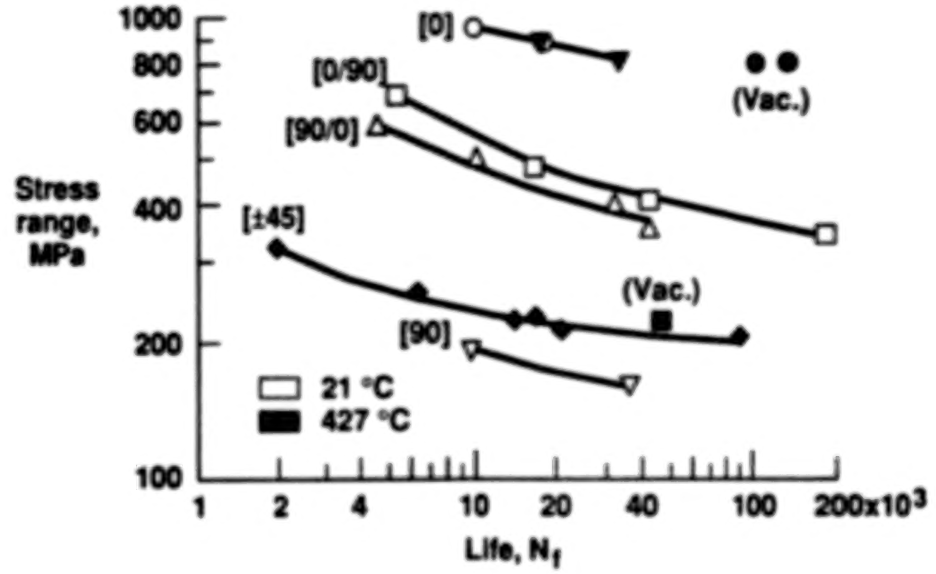
$NbAl_3$

- Patented alloy (Nb-40Al-8Cr-1W-1Y) has oxidation resistance at 1200 °C comparable to Al_2O_3-forming Ni-base alloys.
- AlNbCr is an identified phase that provides oxidation resistance.
- Yttrium is essential for optimum behavior.

CD-91-[illegible]

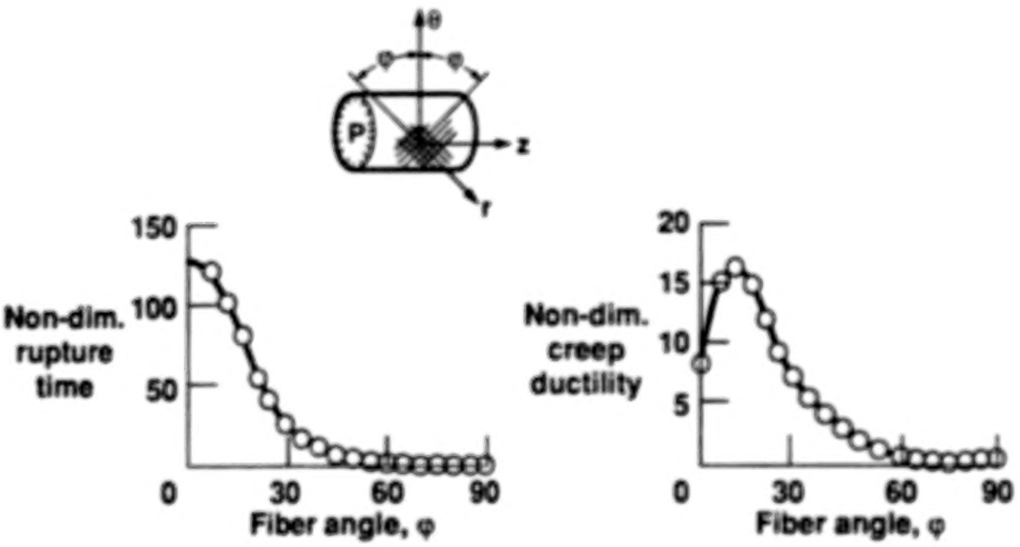
Another key issue for IMC's is the limiting oxidation use temperature of the matrices. Cyclic oxidation behavior of the primary matrices being investigated in HITEMP have been characterized, models developed to predict their behavior, and their oxidation limiting use temperatures defined. Current emphasis is on coatings to further extend the life of these matrices and the composites.

Isothermal Fatigue Life Curves for Various Architectures of 35 vol% SiC/Ti–15–3 Composite



At the lamina level, isothermal fatigue life curves have been experimentally determined for SiC/Ti-15-3 which will be used in verifying analytical models and defining fiber architectures for specific applications. Per given stress range, the $[0]_8$ specimens have the longest life, followed by the $[0/90]_{2S}$, the $[90/0]_{2S}$, the $[\pm 45]_{2S}$, and the $[90]_8$ orientations, respectively. For the few specimens tested, no difference in the fatigue life was observed between the $[0]_8$ specimens tested at 427 °C or those tested at room temperature.

Creep and Creep Rupture of MMC's: Rings



CD-91-[illegible]

Designers want long life and high creep damage tolerance (ductility) in MMC structures (e.g., rings). A recent creep/creep-rupture model for MMC's serves to achieve optimal fiber configurations in creep limited structures.

Intermetallic Matrix Composites

Future research directions

- Continue effort to identify and fabricate high CTE fibers.
- Experimentally verify analytical models for compensating coatings to overcome CTE mismatch.
- Develop fiber coatings for either reaction prevention or bonding enhancement.
- Improve ductility/toughness of matrix candidates through ductile wire toughening concepts.
- Develop reliable oxidation resistant coatings.
- Develop composite models to predict deformation and service life.

CD-91-[illegible]

The issue of fiber-matrix CTE mismatch will continue to plague IMC's until fiber and/or fiber-coating systems are developed that minimize the mismatch to a permissible limit. Coatings may also be required to optimize the fiber-matrix bond. Ductility/toughness of the matrix is another key issue being approached by analytical modeling and experimentally via microalloying, macroalloying, and ductile second phase toughening.

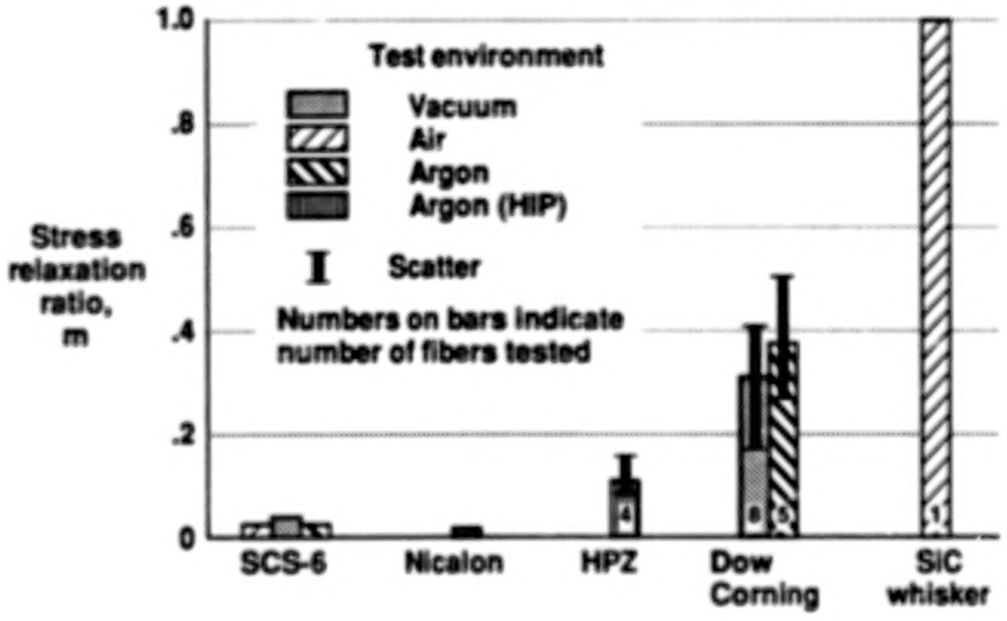
Ceramic Matrix Composites

Key issues

- Strong, small diameter fibers with thermal stability and creep resistance above 1400 °C
- Weak fiber/matrix interfaces with thermo-oxidative stability
- Oxidation resistance matrices with high fracture strength
- Identification of deformation and damage mechanisms and accurate modeling for reliable CMC structures
- Test methodologies for evaluating CMC structural and environmental performance at high temperature

CD-91-[illegible]

Structural ceramic matrix composites are being explored to provide revolutionary advances in the maximum operating temperature of gas turbine engines. A critical issue for these materials is to develop a high strength, small diameter fiber with thermal stability and creep resistance over the temperature range of 1300 to 1650 °C (2375 to 3000 °F).

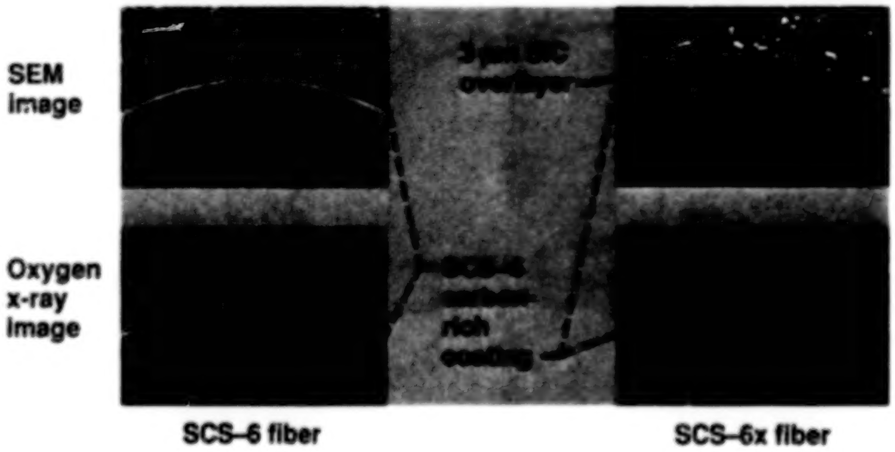


Fiber Comparison at 1400°C
1-hr Stress Relaxation Ratio

A simple measure of creep resistance via a bend test at 1400 °C (2550 °F) has provided a ranking of the creep strength of several fibers. For this bend test, the stress relaxation ratio (as-received/after 1400 °C heat treatment) decreases from unity as the fiber creeps. The most creep resistant fibers evaluated to date are single crystal silicon carbide whiskers. For polycrystalline fibers, creep resistance improves with freedom from silicon and oxygen and with increasing grain size. However, larger grained fibers also display lower tensile strengths.

New Coating for SCS–6 Fibers Improves Oxidative Stability

Interface in SCS–6/RBSN composites after 100 hr, 600 °C in air

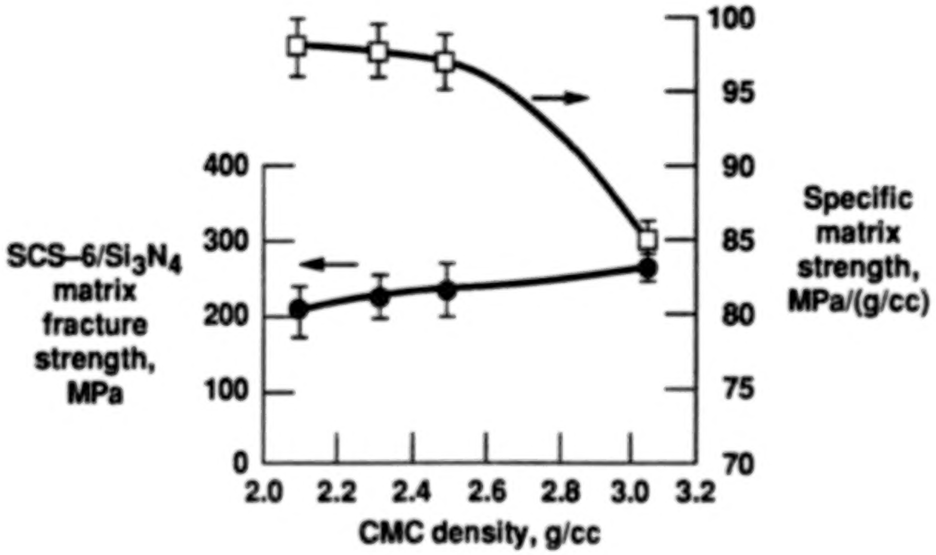

SEM image
Oxygen x-ray image
3-µm SiC overlayer
SCS-6 carbon-rich coating
SCS–6 fiber
SCS–6x fiber

CD-91-54309

A second key issue with CMC's is the fiber-matrix interface, where a weak, thermo-oxidative resistant interface is desirable. Textron has recently developed an SiC overlayer approach which significantly reduces oxygen attack of the SCS-6 fiber. This new fiber, SCS-6x, has the potential for use in porous/cracked matrices as well as in matrices with carbon-attacking compositions, such as oxides, metals, and intermetallics.

ORIGINAL PAGE IS
OF POOR QUALITY

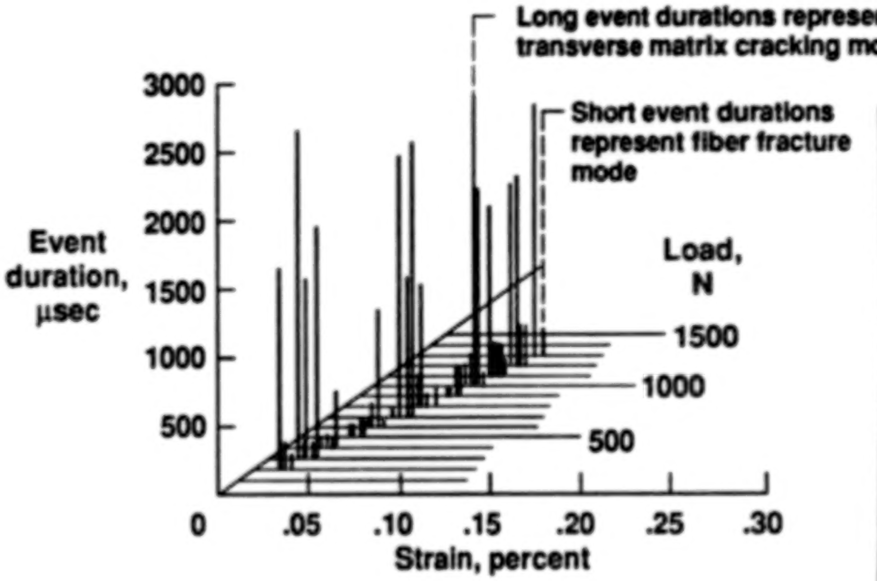
Porous Matrix CMC Can Display Better Specific Strength Than Dense Matrix CMC



CD-91-84310

Matrix fracture strength is a critical issue for the use of CMC's in gas turbine engine applications. Although the strength of SiC SCS-6/RBSN shows a slight increase with increasing density of the composite, on a specific strength basis the low density, porous material exhibits superior strength. This implies that the need for high density CMC's may be reduced and widens the fabrication opportunities for high strength composites.

Damage Accumulation and Failure Mechanisms in SiC/RBSN



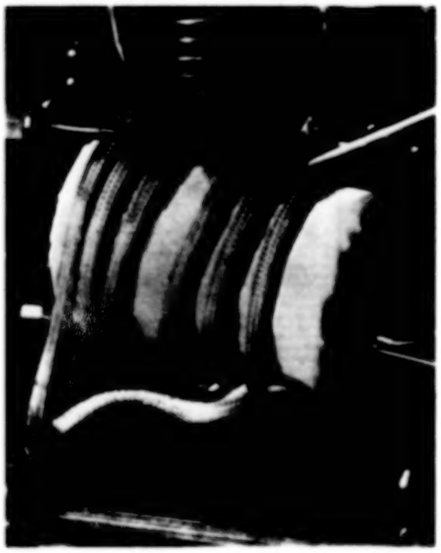
X-radiography results at failure

CD-81-84311

Identification of deformation and damage mechanisms is a critical issue in the development of modes to predict the life of CMC's. Acoustic Emission (AE) has been used in conjunction with x-radiography to detect both matrix and fiber cracking of SiC/RBSN composites tested in tension. The AE results have been confirmed by radiography conducted at different load levels.

ORIGINAL PAGE IS
OF POOR QUALITY

Susceptor Design Used with a 5-kW Induction Heater for Testing CMC's to 1650 °C (3000 °F)



CD-91-54312

The proposed extreme use temperatures of CMC's require the development of high temperature laboratory test equipment that will permit accurate characterization of these materials. Test apparatus has been developed that will permit testing of CMC's in tension and fatigue up to 1650 °C (3000 °F).

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Ceramic Matrix Composites

Future research directions

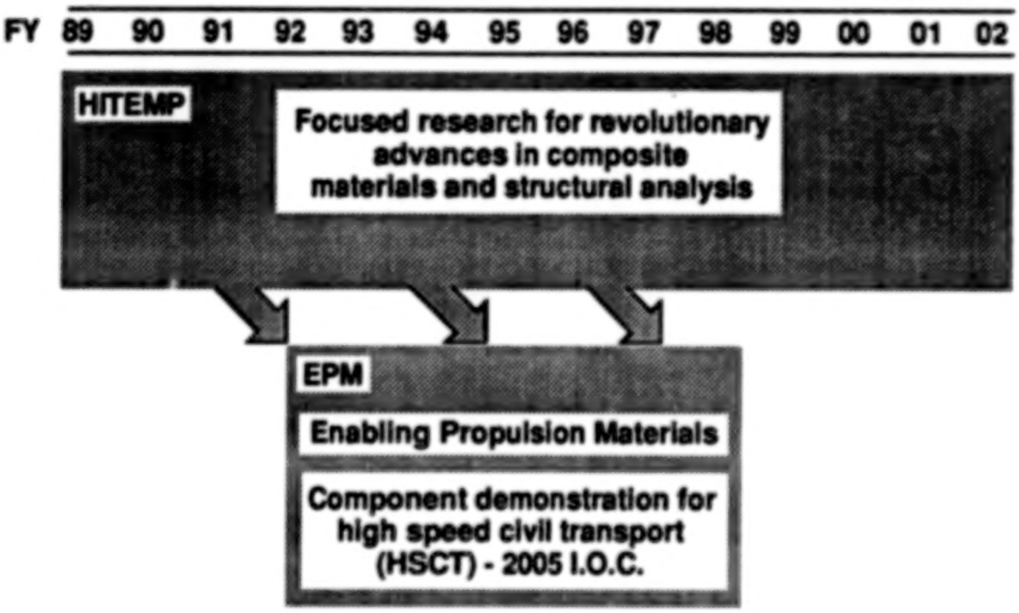
- Define thermomechanical performance limits for polycrystalline and single crystal SiC fibers and single crystal alumina fibers.
- Explore compositions and processing approaches for
  - Improved single crystal oxide fibers
  - Stable CMC interfaces
  - High thermal conductivity Si-based CMC.
- Demonstrate environmental and structural durability limits for high potential CMC.
- Test methodologies for evaluating CMC structural and environmental performance at high temperature.

CD-91-54313

To extend the potential use temperature of CMC's, future research will continue to focus on the development of SiC and Al_2O_3 fibers. Other areas of research will focus on stable interfaces, high thermal conductivity Si-based CMC's, and demonstration of the environmental and structural durability of the most promising materials.

Concluding Remarks

HITEMP future research directions



FY 89 90 91 92 93 94 95 96 97 98 99 00 01 02

HITEMP

Focused research for revolutionary advances in composite materials and structural analysis

EPM

Enabling Propulsion Materials

Component demonstration for high speed civil transport (HSCT) - 2005 I.O.C.

CD-91-54214

Transfer of the technology generated in HITEMP hopefully will see fruition in programs such as IHPTET and in the new Enabling Propulsion Materials (EPM) Program. The major areas of emphasis in the EPM Program for the high speed civil transport will be the combustor liner which will require the application of CMC's to minimize NO_x and the exhaust nozzle where IMC's will be required because of their high strength-to-weight ratio potential.

N91-20111

COMPUTATIONAL MATERIALS SCIENCE AN EXAMPLE: NUMERICAL MODELING OF CHEMICAL VAPOR DEPOSITION PROCESSING OF ADVANCED FIBERS

Arnon Chait, Suleyman Gokoglu, and Maria Kuczmarski
NASA Lewis Research Center
Cleveland, Ohio

Pang Tsui
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park, Ohio

Lisa Veitch
NASA Lewis Research Center
Cleveland, Ohio

David Thompson* and Ronald Gaug*
The University of Akron
Akron, Ohio

Computational materials science is a rapidly emerging discipline combining aspects from chemistry, fluid dynamics, and materials and computational sciences. This paper will detail the initial efforts undertaken in the Materials Division at Lewis to establish a dedicated facility for numerical and analytical simulations of materials processing. Computational Materials Laboratory activities are currently aimed at understanding and optimizing processes involving transport of heat, fluid, and mass. An ongoing combined experimental and numerical study is presented. It illustrates the growth of advanced fibers for high-temperature composite applications by using the chemical vapor deposition process. A comprehensive analysis of the three-dimensional heat and fluid flow inside the reactor along with simultaneous computations of the chemically reacting gas and fiber surface species are shown to provide an enhanced level of understanding of the process. The difficulties in simulating the entire process, as well as the present accomplishments, are discussed, and future directions are suggested.

---

*NASA Resident Research Associate at Lewis Research Center.

Mission

- **Apply existing computational techniques to materials science and develop new ones in order to**
  - **– Understand and improve processes**
  - **– Facilitate design of efficient experiments**
  - **– Develop and test advanced processing concepts with emphasis on optimization**
- **Provide *easy and informal* access to advanced computational technology in materials science to U.S. industry and universities**
- **Focus on advanced high-temperature materials for NASA programs such as HiTEMP and the High Speed Civil Transport**

CD-91-54150

The Computational Materials Laboratory represents a new venture for the Materials Division at Lewis. Advances in computational sciences and computer hardware have enabled a complementary avenue of research in materials science. The initial efforts in the computational laboratory are concentrated in areas of largest payoffs: space and high-temperature materials processing. The emphasis is on practical and timely solutions to a wide range of real world problems. A relatively unique aspect of the laboratory is the informal and easy access it offers to a variety of organizations that need to develop, understand, or optimize materials processes.

Why Modeling?

Space processing	Earth processing
Few, unique, very expensive experiments	Many, similar, relatively inexpensive experiments
Need better understanding of fundamentals	Cut lead time of new materials development
Very high cost	Very high cost

Modeling can

Enhance level of fundamental understanding, thereby leading to

- Shorter development time
- Cost-effective and timely solutions

CD-91-54191

Both space and earth-bound processing require large investments in time and money. Although space experiments are few and unique, they are individually very expensive, and the limited access to space prohibits the usual iterative experimental design. Earth processing involves relatively inexpensive experiments, but the total number needed for the development of new materials or processes usually results in a very long lead time and a large investment. A directed modeling program in tandem with the traditional experimental approach can cut the development time and lead to a greater understanding of the process.

Underlying Physical Phenomena in Materials Processing

	Directional solidification	Chemical vapor deposition
Usage	Ranges from semiconductors to gas turbine blades	Composite fibers, tailored surface coatings, optical fibers
Process parameters	Thermal profile	Thermal profile
Basic phenomena	Heat and mass transport due to density gradients	Heat and mass transport due to density gradients
Unique characteristics	Phase change boundary	Chemical reactions, radiation heat transport
Mathematical description	Navier-Stokes equations	Navier-Stokes equations

CD-91-54152

The primary reason for the success of the computational approach is the similarity in the underlying physical phenomena among seemingly different processes. For example, most processes are based on phase transformation from either the gas or liquid states to the solid state; and processes are typically governed by an intricate balance of heat and mass transport, and possible chemical reactions. The mathematical description of transport phenomena from a macroscopic viewpoint is provided by the Navier-Stokes equations. Therefore, fairly generic computational techniques developed elsewhere can be employed in the study of similar processes. Only relatively minor modifications are usually needed then after an initial investment in describing one such process.

Current Studies

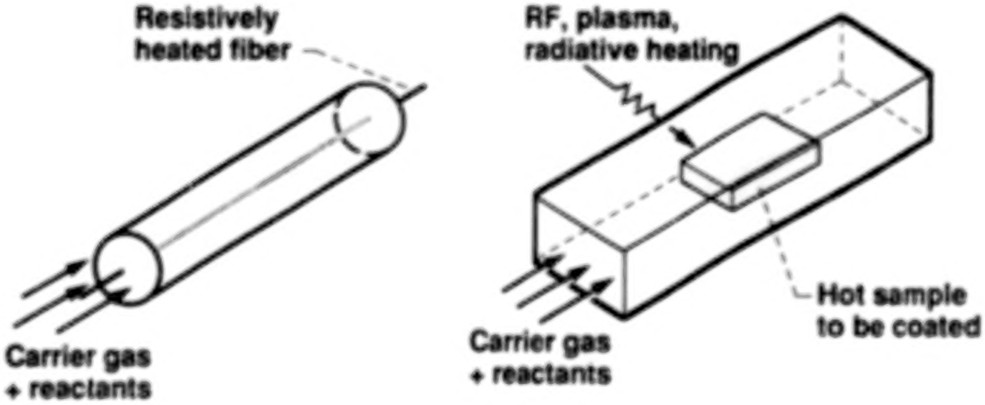
- **Solidification processes**
  - **Directional solidification**
    - Space-bound experiments
    - Transient effects
    - Furnace-sample coupling and optimization
  - **Zone and float zone**
    - Asymptotic/numerical analysis
    - Core fiber float zone
  - **Fundamentals**
    - Phase transition formulation
    - Free boundary problems – a generalized numerical algorithm
    - Interfacial morphology and pattern formation
    - Applications
      - –thin-film high Tc superconductors
      - –free surface convection
- **Gas phase processes**
  - **Physical vapor deposition**
    - Single crystal opto-electronic
  - **Chemical vapor deposition**
    - Si, C, and SiC CVD
    - Fibers and surface coatings
    - SiC high temperature semiconductors
- **Other processes** →
  - Thermal energy storage materials
  - Molecular modeling
  - Plasma-based process modeling
  - Solar cell parameterization

CD-91-54153

The initial emphasis in the Computational Materials Laboratory reflects a heavy investment by the space community, where modeling contributions have already been realized. At this stage we are increasing our effort in the study of high-temperature materials. The chemical vapor deposition (CVD) process is being intensely investigated in a combined experimental/computational study. Future work will be directed at process optimization, and at modeling of processes strongly affected by transport phenomena, both diffusion and convection.

Chemical Vapor Deposition

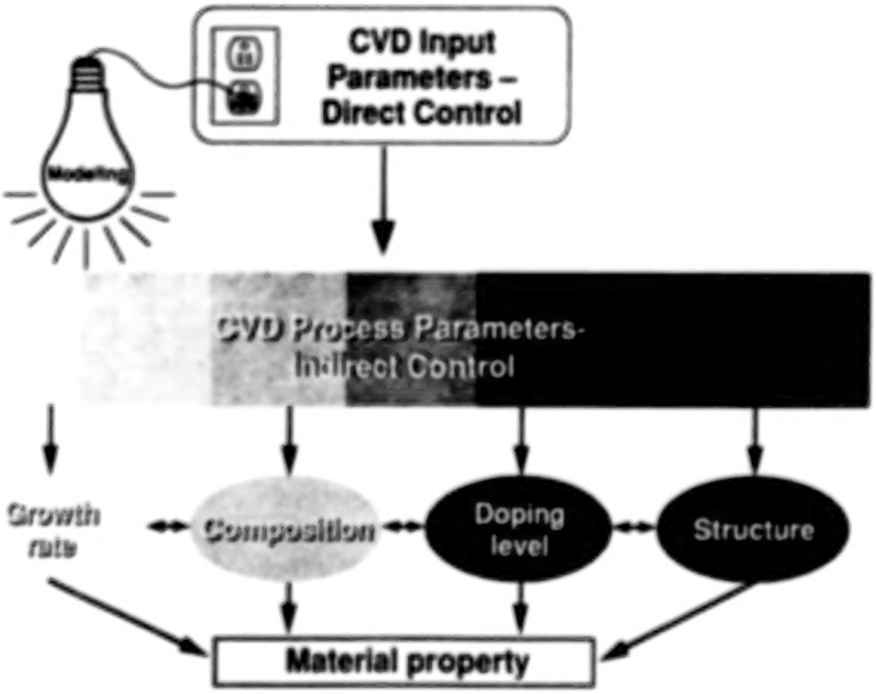
- Versatile technique, easily tailored for
  - Fiber growth and coating
  - Environmental coating and infiltration


Resistively heated fiber
Carrier gas + reactants
RF, plasma, radiative heating
Hot sample to be coated
Carrier gas + reactants

- Process involves
  - Transport of heat and mass in the gas stream
  - Simultaneous chemical reactions in gas stream and on the sample surface

CD-91-54198

The CVD process is a general purpose, efficient technique for such diverse applications as semiconductor film growth, composite materials fiber growth and coating, and environmental coating including bulk infiltration. In this process a sample heated via RF, joule-heating, or other method is placed in a reactor and exposed to a flow of carrier and chemically reacting gases. Subsequent simultaneous gas and surface reactions result in the growth of a surface layer of a tailored chemical composition and, sometimes, microstructure. Because of its versatility and relative simplicity, the CVD process is growing in popularity. At Lewis it is applied for high temperature semiconductor work (SiC), for fiber production (SiC, other), and for environmental durability coatings.


CVD Input Parameters – Direct Control
Modeling
CVD Process Parameters- Indirect Control
Growth rate
Composition
Doping level
Structure
Material property

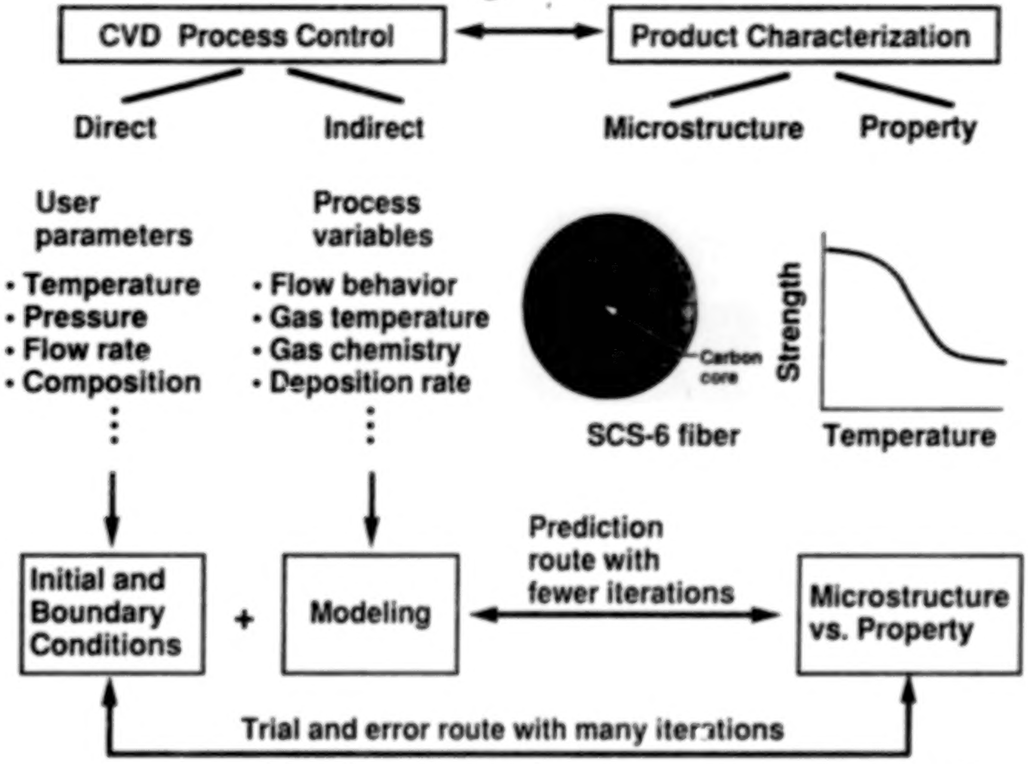
CD-91-54157

The CVD process, although simple to describe, comprises a very complex combination of simultaneous phenomena which interact to form a growing structure on the surface. The problem of designing and optimizing the process is common to all processes: How do we alter the process *output* parameters (growth rate, composition, doping, and microstructure) when we have direct control of the *input* parameters only?

This problem can, in principle, be addressed via several approaches, including a "black box" process optimization methodology. Treating the process as a whole, and using statistical and other optimization tools, can provide relatively quick results, but this does not further our understanding of the process. Furthermore, such action does not provide a clear path for process scaleup; a research process that is optimized on the laboratory scale generally cannot be directly translated into a production scale by using process optimization alone. Only a process-motivated fundamental analysis can, in the long run, provide the knowledge required for process alteration or scaling.

The present state of the art in fundamental CVD modeling allows researchers to understand the relations among the process input parameters that lead to certain growth rate characteristics. We also are beginning to correlate those variables with surface composition, but present day models still cannot adequately explain the resulting microstructure. Nevertheless, even prediction and control of growth rates may offer substantial benefits to process designers, and can sometimes be well-correlated with the microstructure.

Role of Modeling in CVD Processes



CVD Process Control
Product Characterization
Direct
Indirect
Microstructure
Property
User parameters
• Temperature
• Pressure
• Flow rate
• Composition
Process variables
• Flow behavior
• Gas temperature
• Gas chemistry
• Deposition rate
Carbon core
SCS-6 fiber
Strength
Temperature
Initial and Boundary Conditions
+
Modeling
Prediction route with fewer iterations
Microstructure vs. Property
Trial and error route with many iterations

CD-91-54158

In analyzing the CVD process, we distinguish among direct and indirect parameters. The former refer to variables which are controlled by the user, whereas the latter are the result of the fluid, heat, and mass transport combined with simultaneous chemical reactions in the reactor. Fundamental process analysis helps explain how the direct parameters result in an indirect process behavior. Such knowledge can result in fewer iterations in the laboratory and a more rapid transfer of a laboratory experiment into a production process.

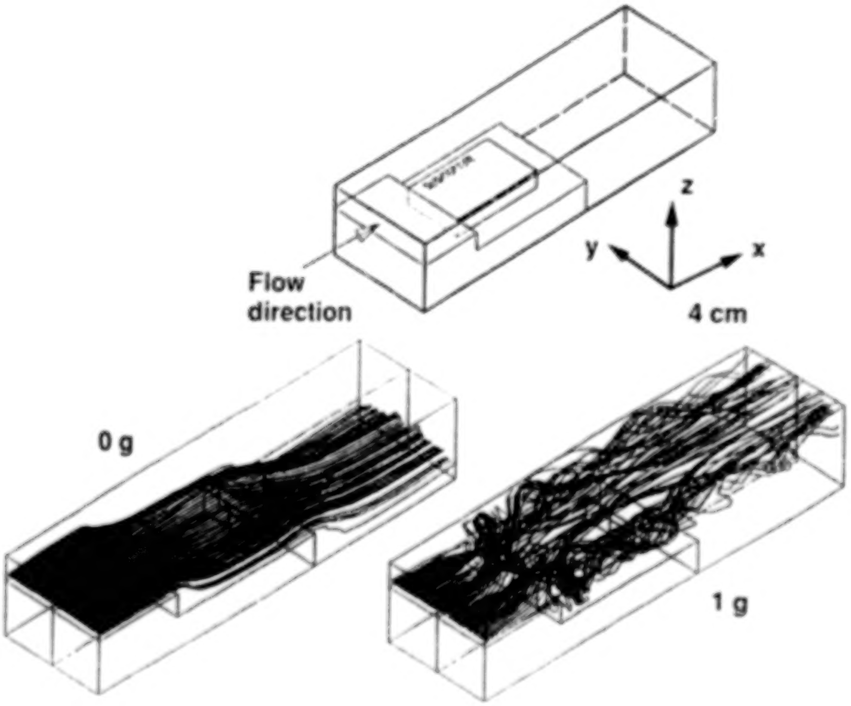
General Characteristics of Lewis CVD Models

- Three-dimensional steady momentum, heat, and mass transport
- Variable properties, including density for nondilute systems
- Secondary transport effects of significance to CVD: Söret diffusion
- Finite rate chemical kinetics in the gas phase
- Finite rate chemical kinetics on the surface, including sticking probabilities

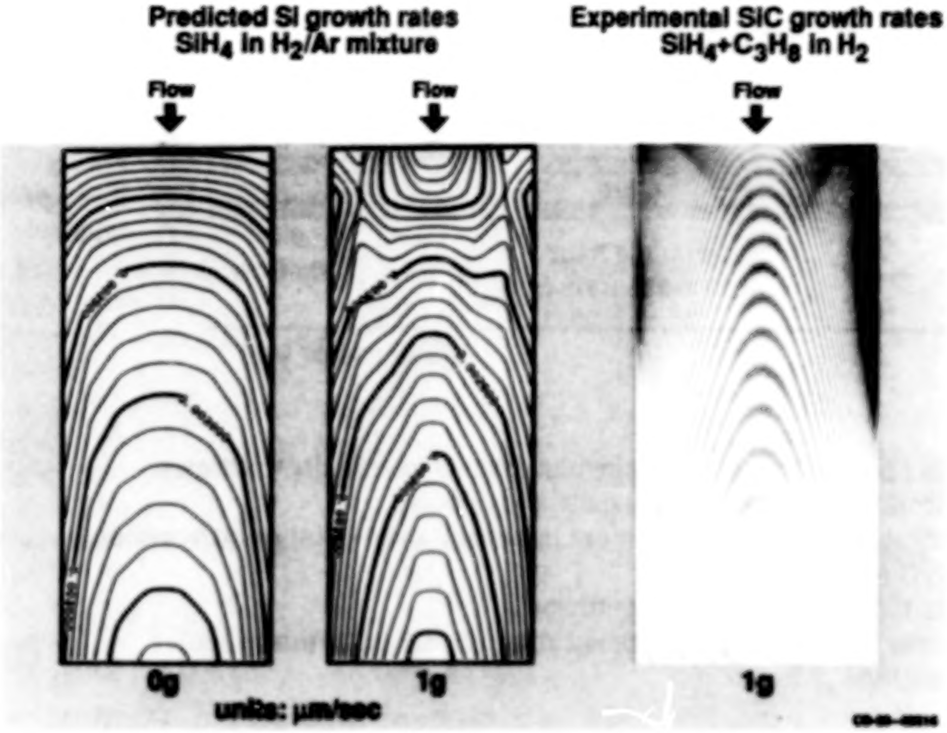
CD-91-54159

The modeling codes for CVD applications at Lewis are based on modified general purpose computational fluid dynamics codes. These codes, which permit arbitrary three-dimensional geometries, solve for the fluid, heat, and mass transport in the reactor, and the simultaneous chemical reactions in the gas and on the surface. The chemistry models include finite rate reactions, allowing for realistic kinetic effects. Other secondary transport mechanisms, such as Soret transport of mass due to thermal gradients (in contrast to the common Fickian diffusion of mass due to concentration gradients), are also included. These second-order effects can become the predominant mechanisms in fiber growth if very steep thermal gradients are present in the vicinity of the fiber surface.

Modeling of Si CVD Coating

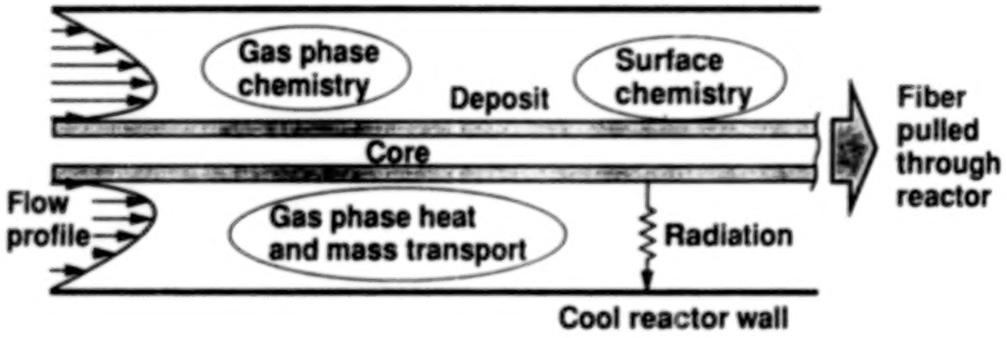


An initial model development and verification study was directed at simulating a well-documented CVD experiment, Si coating of a flat graphite susceptor. The rectangular channel reactor was water cooled, and the sample was heated by an RF field. A complex, three-dimensional flow field resulted from a combination of natural convection originating from the strong density gradients (due to thermal gradients between the hot sample and the cooled walls) and the forced flow of gases through the reactor. Fundamental models allow researchers to explore the origins of transport patterns in the reactor. The simulation shown on the left explores the role of buoyancy driven convection by artificially setting the gravity level to zero, thereby eliminating buoyancy altogether. The flow through the reactor (visualized by following traces of massless particles injected into the reactor) is much more uniform without the corkscrew-like convective rolls that caused complex three-dimensional temperature and chemical species fields, and ultimately, nonuniform surface deposition.



The computed deposition rate contour maps for the previous figure are shown in the left two plots. Both the appearance of the contours and the rates of deposition (in one g) closely correspond to the laboratory measured values. A comparison to the zero-g case (on the left) elucidates the deleterious effects of the corkscrew convection resulting from buoyancy. It is interesting to note that the contour map of the laboratory grown SiC film qualitatively resembles those of the computed patterns, even though the computed chemistry did not consider the carbon species, and the reactor geometry and operating conditions were not identical for both cases. For this particular configuration both of the processes were limited by the transport of chemically reacting species to the surface; therefore, similar growth rate contours resulted.

Fiber CVD

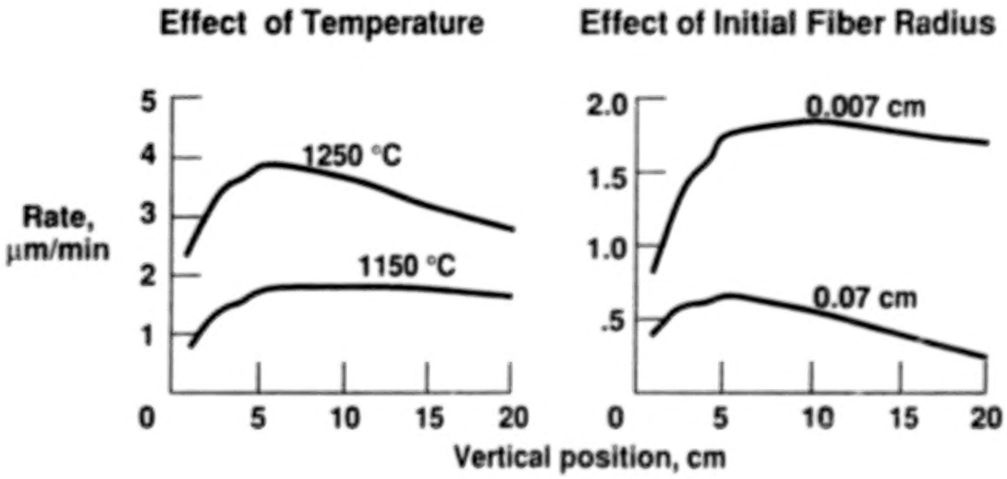

Gas phase chemistry
Surface chemistry
Deposit
Core
Fiber pulled through reactor
Flow profile
Gas phase heat and mass transport
Radiation
Cool reactor wall

- Gas phase and surface chemistries are practically unknown
- Growth rates vary both axially and radially
- Söret diffusion is much more important than in other conventional CVD reactors
- Radiation effects are significant
- Fiber temperature control is extremely difficult with resistive heating

CD-91-54162

Chemical vapor deposition modeling of SiC or other fiber growth is easier in one respect than the previously shown case, but it is also more complex because of additional nontrivial aspects. Although the cylindrical geometry lends itself to easier model description (axisymmetric versus three-dimensional), the effects of secondary transport modes (Soret) become dominant and require very fine computational meshing near the fiber. Additionally, a common problem for all computational chemistry codes is the lack of reliable chemical kinetics data, especially for exotic Si and C carriers. At present, this last difficulty represents the most important challenge yet to be overcome.

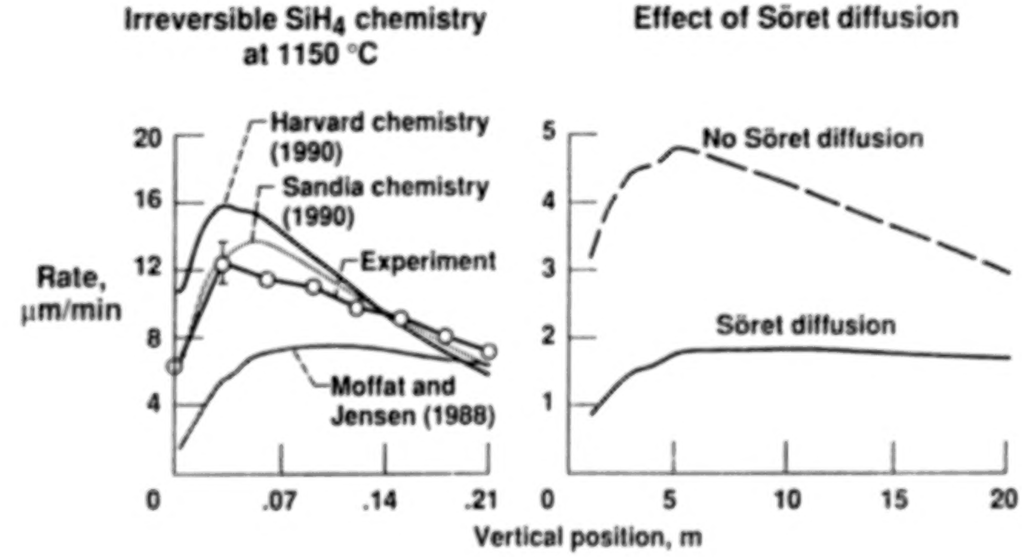
Processing Parameters Effects on Deposition Rates and Uniformity



CD-91-54163

These plots show the level of practical understanding gained from fundamental fiber CVD modeling. The plot on the left shows axial deposition rate distributions (i.e., along the fiber from inlet to outlet) as functions of fiber temperature. The point to be stressed here is that a 100-°C difference in fiber temperature produces a two-fold difference in deposition rate. The more rapid deposition at the elevated temperature causes subsequent depletion of nutrients in the reactor thereby resulting in uniformly decreasing deposition rates along the fiber. The lower temperature setting, although producing lower instantaneous deposition rates, does not deplete the nutrients in the reactor, so very uniform rates are established once the initial transient heating stage is passed. From an operational viewpoint, a 100-°C control of a growing fiber which is resistively heated can be very difficult because of the constantly changing fiber resistivity during the growth. The feedback and process sensitivity to direct parameter settings are clearly understood from this example of results available from fundamental process modeling. Similarly, other "what if" questions can be answered easily with a model. For example, the surface area of a larger initial fiber causes an increase in the deposition rates, as shown on the right plot.

Subtle Effect on Deposition Rates
Need for Fundamental Understanding



Irreversible SiH_4 chemistry at 1150 °C

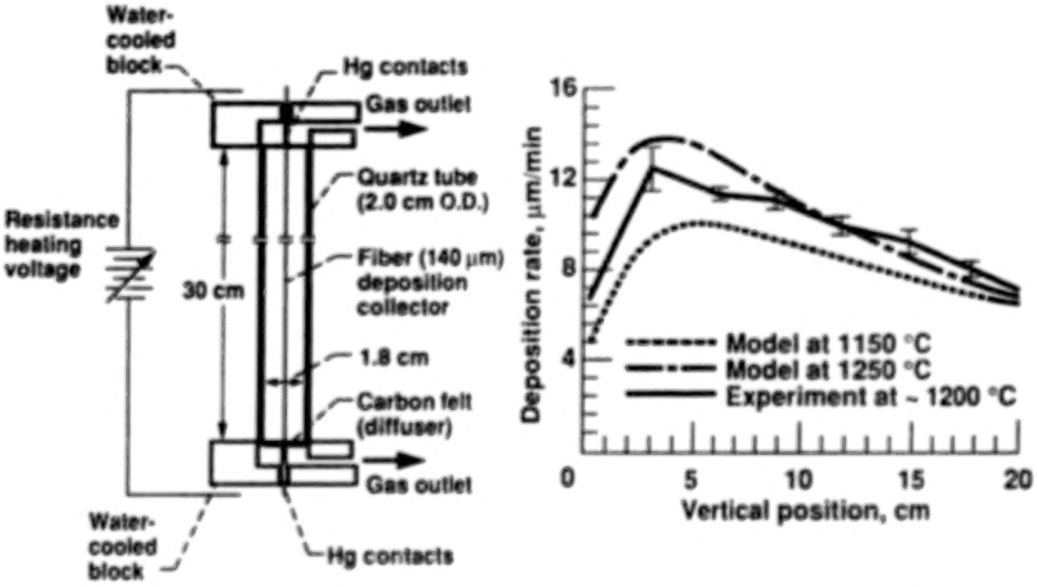
Effect of Söret diffusion

CD-91-54164

The requirements for a successful fundamental process modeling are highlighted in the two plots shown in this figure. The left graph underscores the uncertainties involved in using different chemical kinetics data obtained from recent literature. It is clear that an ad-hoc reliance on published data is unwarranted even for this well-studied system. More complex chemical molecules will result in even more difficult chemistries. Clearly, research in basic computational chemistry and detailed kinetics measurements are needed.

The plot on the right is a rather dramatic example of the effect of Soret diffusion on the process. Soret diffusion is the separation of species of different molecular weights because of steep thermal gradients. This effect is traditionally thought to be of secondary importance for most applications, but in fiber growth, where the radial temperature gradients in the very near vicinity of the fiber might be of the order of 10^4 to 10^5 °C/cm, the Soret effect becomes the dominant mechanism in the transport of species to the fiber and must be included in any viable fiber CVD model.

Comparison of CVD Model Predictions With Experimental Measurements for Si Deposition Rates



Schematic of experiment

Model vs. Experiment

CD-91-54165

An experimental verification of deposition rates along the fiber in a specially built reactor has shown good agreement with modeling results, both qualitatively and quantitatively. Future use of the model is directed at providing guidance for CVD fiber reactor design and operation, with the goal of producing uniform deposition rates.

Conclusions

- Computational materials science is an emerging discipline with a potential for
  - Enhancing our level of understanding of materials processes
  - Accelerating development of new materials

- The Materials Division has established a dedicated Computational Materials Laboratory for simulating advanced materials processing with an emphasis on timely and relevant solutions for processes involving
  - Phase change
  - Transport of and reactions among chemical species
  - Complex fluid, heat, and mass transport
  - Process optimization

CD-91-[illegible]

N91-20112

OVERVIEW OF ROTORCRAFT AND GENERAL AVIATION PROPULSION TECHNOLOGY

John J. Coy and Joseph D. Eisenberg
NASA Lewis Research Center
Cleveland, Ohio

This paper presents an overview of NASA'S propulsion research that is aimed at applications for rotary winged flight and general aviation. The strategic goal of this research effort is to provide innovative technologies that will strengthen the nation's competitive stance in the world market for the civil sector, and also provide superior rotorcraft for U.S. military use.

NASA and the U.S. Army work hand-in-hand on rotorcraft propulsion research at NASA Lewis Research Center. This is made possible by a cooperative NASA-Army agreement which has situated the Army's center for basic propulsion research at Lewis.

The NASA Lewis tactical plan for achieving this strategic goal is to (1) reduce fuel consumption of small engines by 30 percent through use of advanced cycles (including higher pressure and temperature operation) and with improved turbomachinery components technology and ceramic materials, (2) contribute to fuel savings through weight savings and to reliability by developing advanced technology for transmissions, (3) contribute to aircraft safety by providing advanced anti-icing and deicing technology,[1] and (4) achieve high-speed capability through advanced propulsion systems.

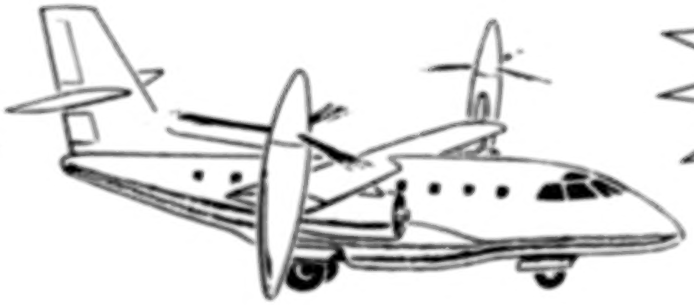
In this session of the conference, the work efforts that are directed at meeting these challenges are reviewed. In this paper, high-speed propulsion studies aimed at enabling 450-kn rotorcraft are reviewed.

Subsequent papers review the following focused projects: (1) rotary engine technology that will achieve multifuel capability, a power density of 5 hp/in.^{3} engine displacement, and a specific fuel consumption of 0.35 lb/hr/hp, and (2) advanced rotorcraft transmissions that reduce weight by 25 percent, reduce gear noise by 10 dB, and increase mean time between removal to 5000 hr. Also reviewed is the base research and technology effort in turbomachinery for small engines. This includes research directed at compressor technology achieving 24-to-1 pressure ratios and 83 percent polytropic efficiency, combustor technology achieving 3000 °F capability and more uniform combustor exit temperature distributions as described by a 0.1 pattern factor, and turbine technology achieving 88- to 90-percent adiabatic efficiency and cycle temperatures of 2600 °F.

---

[1] The Aircraft icing technology is presented in the session on Multidiscipline Research.

Rotorcraft and General Aviation



- Fuel economy
- Safety
- Reliability

Systems study
- High-speed propulsion systems

Focused projects
- Rotary engine
- Advanced transmission - ART

Base R&T
- Compressors
- Combustors
- Turbines
- Transmissions

CD-91-54045

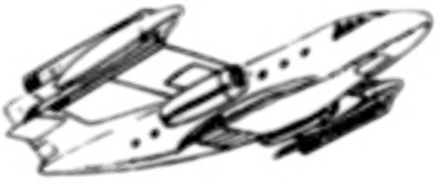
This paper presents an overview of NASA'S propulsion research that is aimed at applications to rotary winged flight and general aviation. The tactical objectives of the program are to (1) reduce fuel consumption of small engines by 30 percent, (2) increase fuel savings and reliability by advanced technology for transmissions, and (3) improve aircraft safety by providing advanced anti-icing and deicing technology.[2]

In this session of the conference, the work efforts shown in this figure are reviewed. To meet the goals of fuel economy, safety, and reliability, there is continuing base research and technology in small engine turbomachinery. These goals will be achieved through advanced engine cycles with higher pressure and temperature operation enabled by advanced component technology, with applications directed at compressors, combustors, and turbines. There is continuing basic research on transmission technology for lighter, quieter, and more reliable gearboxes. These base research and technology efforts are presented in more detail in the following presentations, and subsequent papers review the following efforts on focused projects: (1) rotary engine technology that will achieve multifuel capability, a power density of 5 hp/in.^{3} engine displacement, and a specific fuel consumption of 0.35 lb/hr/hp, and (2) advanced rotorcraft transmissions that reduce weight by 25 percent, reduce gear noise by 10 dB, and increase mean time between removal to 5000 hr.

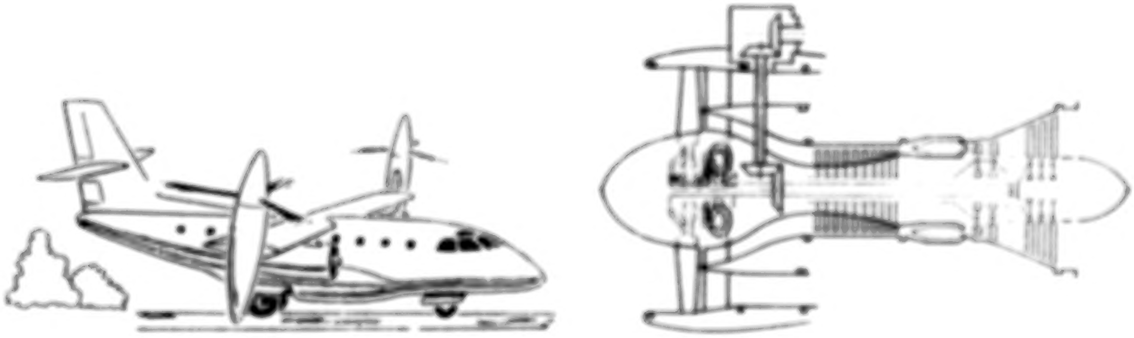
The remainder of this presentation reviews the systems study on high-speed propulsion systems for 450-kn rotorcraft.

---

[2] Aircraft icing technology is presented in the session on Multidiscipline Research.



High-Speed Rotorcraft Propulsion Studies



CD-91-54046

High-speed rotorcraft are those capable of cruise speeds from double those of helicopters up to Mach 0.8. The configurations of such aircraft differ greatly from those of helicopters, and their propulsion requirements, for the most part, are extremely demanding and require novel engine approaches. Studies of these rotorcraft were begun in 1988 and are being completed at the present time.

V-22 Osprey

- Helicopter in vertical mode
- 320-kn turboprop in cruise

CD-91-54047

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

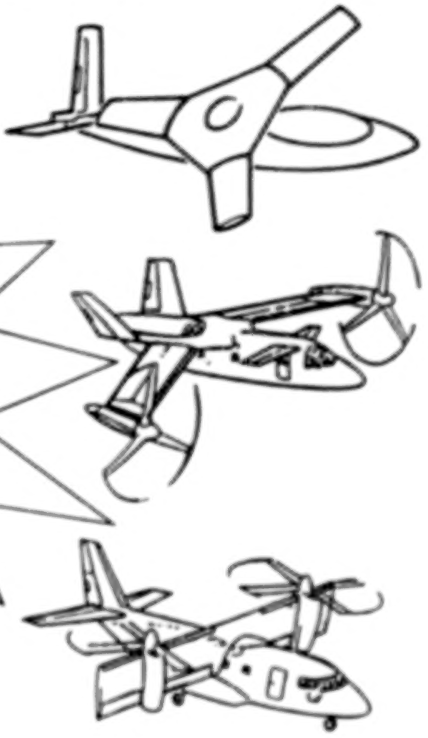
The first mission-capable, high-speed rotorcraft is the V-22 Osprey, a military vehicle capable of a 320-kn cruise speed. The Osprey takes off and lands like a helicopter, has hover efficiency like a helicopter, and can reposition its rotor blades to fly like a turboprop. The Osprey's first flight was on March 20, 1989.

We believe that a similar civilian aircraft could be used to help alleviate the problem of airport congestion without demanding the construction of additional runways. Studies on the application of this type of vehicle have been in progress since 1984 and have considered the global issues of transportation strategy and market strategy, as well as technology barriers. These studies concluded that a good market exists, that there is indeed a place for such a vehicle in the transportation industry, and that civil-specific technology must be developed rather than adapting the military technology. A major issue that has to be addressed is the integration of the civil tilt rotor into the infrastructure of the existing global transportations systems.

These studies are projecting that it is feasible to have a civil tilt rotor transportation system in place within 10 years.

High-Speed Rotorcraft Goals

- 450-kn speed
- Efficient hover
- Low noise
- Affordable
- Flexible
- Maneuverable
- Agile
- Survivable



CD-91-54048

If rotorcraft could increase their speed to 450 kn, they could takeoff and land without runways and could combine efficient hover with the cruise speeds (and range) of current turbofan transports. Thus, the civilian version in the next century would be able to operate, in some instances, not as a feeder aircraft, but rather as the primary mode of transportation. The high speed could also be used advantageously in some military operations. The assumed end-user requirements would be for military attack, military transport, and civil transport versions of a high-speed rotorcraft.

In light of this opportunity, the NASA Ames Research Center has sponsored and directed high-speed rotorcraft studies, beginning in 1988. Their goals, which are listed in this figure, cover both civilian and military requirements.

The Government has encouraged its contractors to consider a great diversity of aircraft concepts in order to explore the maximum possible gains in capability.

High-Speed Rotorcraft Studies

Objectives

- Identify and evaluate concepts
- Identify critical technology
- Define development plans

Approach

- Ames (airframe), Lewis (propulsion)

Contractors

- Bell, Boeing, McDonnell-Douglas, Sikorsky
- Allison, General Electric

CD-91-54049

The objectives of the high-speed rotorcraft studies were to identify attractive high-speed rotorcraft concepts (for both the airframe and propulsion system) and evaluate them against the mission requirements for military attack, military transport, and civil transport. The studies also were to identify the critical technologies that were required for success and that needed further development. They were to lay out development plans that would bring the new vehicles into operation by the turn of the century.

The airframe studies have been funded and directed by Ames Research Center. Coordinated with these studies, but separately funded and directed by Lewis, are the propulsion studies.

The helicopter airframe companies involved are Bell, Boeing, McDonnell-Douglas, and Sikorsky. The propulsion companies are Allison and General Electric.

Space limitations and the propulsion theme of the conference do not permit a complete review of the airframe-related findings of these studies. Briefly stated, weight reduction technology will be a big factor in achieving the goals, and we expect that it will be accomplished through innovative structural design techniques and advanced lightweight materials. The propulsion aspects of the studies are reviewed next.

Propulsion Study Process

(1) Identify concepts

(2) Match to airframe and mission

(3) Scale performance

(4) Identify key technologies

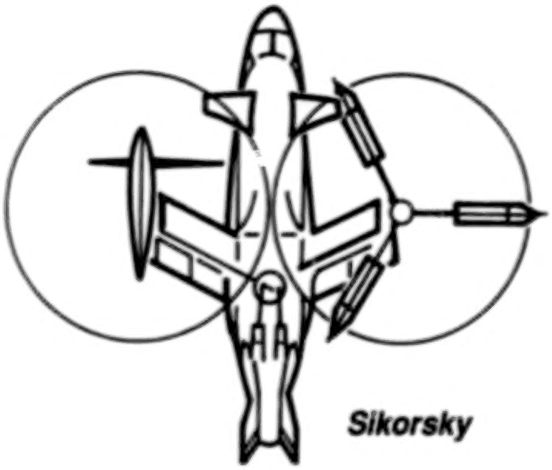
CD-91-54050

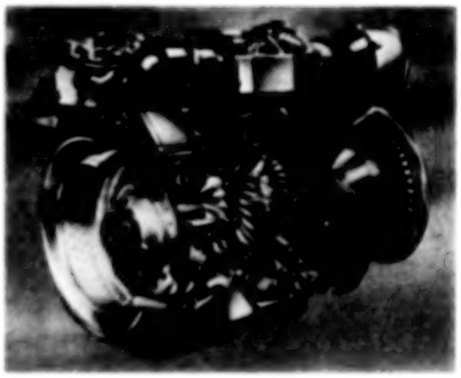
There were four steps in the propulsion study efforts: (1) identify the engine concepts that are suitable for high-speed rotorcraft; (2) from those identified, select the engine concepts best suited for each rotorcraft and mission combination that was identified in the Ames study; (3) for the selected engine concepts, estimate the performance, weight, and other characteristics, and scale the engines over a useful range; and finally, (4) identify the key technologies that need to be developed to have a viable propulsion system for high-speed rotorcraft.

The results of the airframe studies have been examined, and necessary modifications in engine concepts and identification of required technologies have been made. This process has been repeated in an iterative manner in order to determine the most appropriate direction for future work.

Proposed Rotorcraft and Candidate Engines

Variable–Diameter Tilt Rotor





Conventional Turboshaft Engine

CD-91-54051

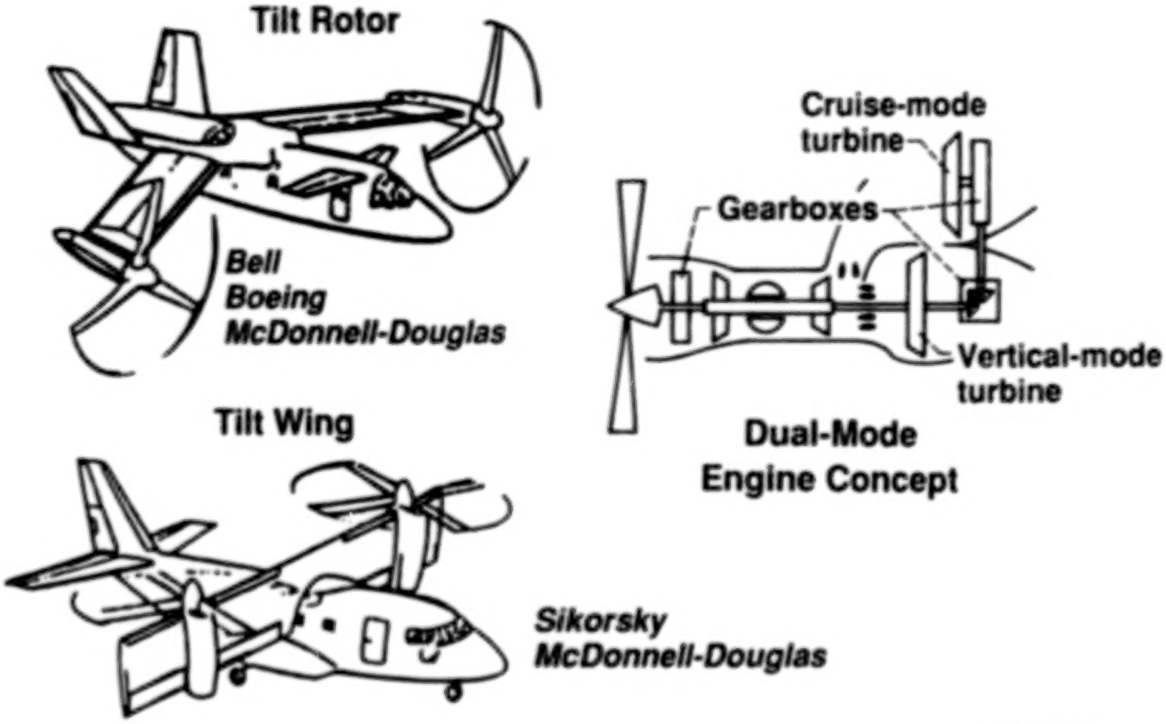
The high-speed rotorcraft airframe studies have defined several interesting vehicle concepts - each dependent on the mission requirements (civil or military). In general, 450 kn was found to be the maximum practical speed - beyond that, the weight penalties began to increase at an ever-increasing rate.

In the next few figures, the aircraft chosen in the high-speed rotorcraft studies for final evaluation are shown along with the specific engine requirements for each.

Shown here is a high-speed tilt rotor aircraft capable of 450 kn. It has highly swept wings and canard control surfaces. To prevent the helical tip speed of the rotor from approaching Mach 1 during cruise, the variable-diameter rotor can be reduced in size for high-speed cruise. The required engine would be a rather standard turboshaft engine, and the only new technology required would be the application of IHPTET GEN 6 (turbine engine) technology to reduce fuel use and weight.

This configuration is a Sikorsky candidate, intended for the military transport and the military scout-attack missions at gross weights of approximately 40 000 and 25 000 lb, respectively.

Proposed Rotorcraft and Candidate Engines


Tilt Rotor
Bell
Boeing
McDonnell-Douglas
Tilt Wing
Sikorsky
McDonnell-Douglas
Cruise-mode turbine
Gearboxes
Vertical-mode turbine
Dual-Mode Engine Concept

CD-91-54052

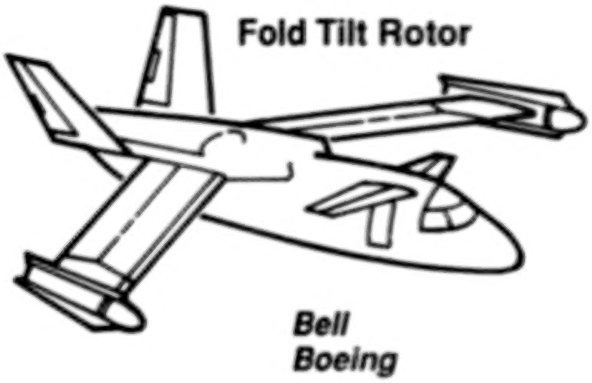
Bell, Boeing, and McDonnell-Douglas investigated similar high-speed tilt rotors with constant-diameter rotors. McDonnell-Douglas and Sikorsky suggested tilt wing aircraft, also with constant-diameter rotors.

Here, propulsion systems capable of operating efficiently at constant power, but with lower rotational shaft speeds at cruise are required to keep the rotor helical tip speed subsonic. A conventional turboshaft engine with a two-speed transmission would be feasible.

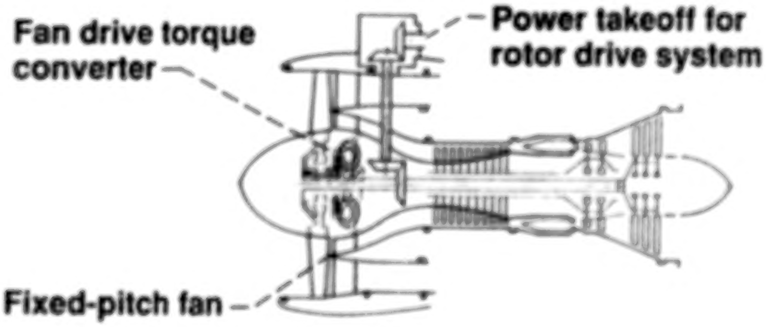
However, the new dual-mode engine concept would not need a two-speed transmission. The configuration illustrated in this figure is merely an example. In the vertical mode, the vanes direct all the flow to the low-speed, vertical-mode turbine. During high-speed cruise, the vanes direct all the flow to the cruise-mode turbine, which would rotate at design speed for maximum efficiency. The gearboxes reduce the speed to drive the rotor blades at their most efficient speed.

The actual definitions of efficient dual-mode concepts are still being developed. Since such engines do not exist now, the technological work required is expected to be significant.

Proposed Rotorcraft and Candidate Engines


Fold Tilt Rotor
Bell
Boeing

Convertible Engine


Fan drive torque converter
Power takeoff for rotor drive system
Fixed-pitch fan

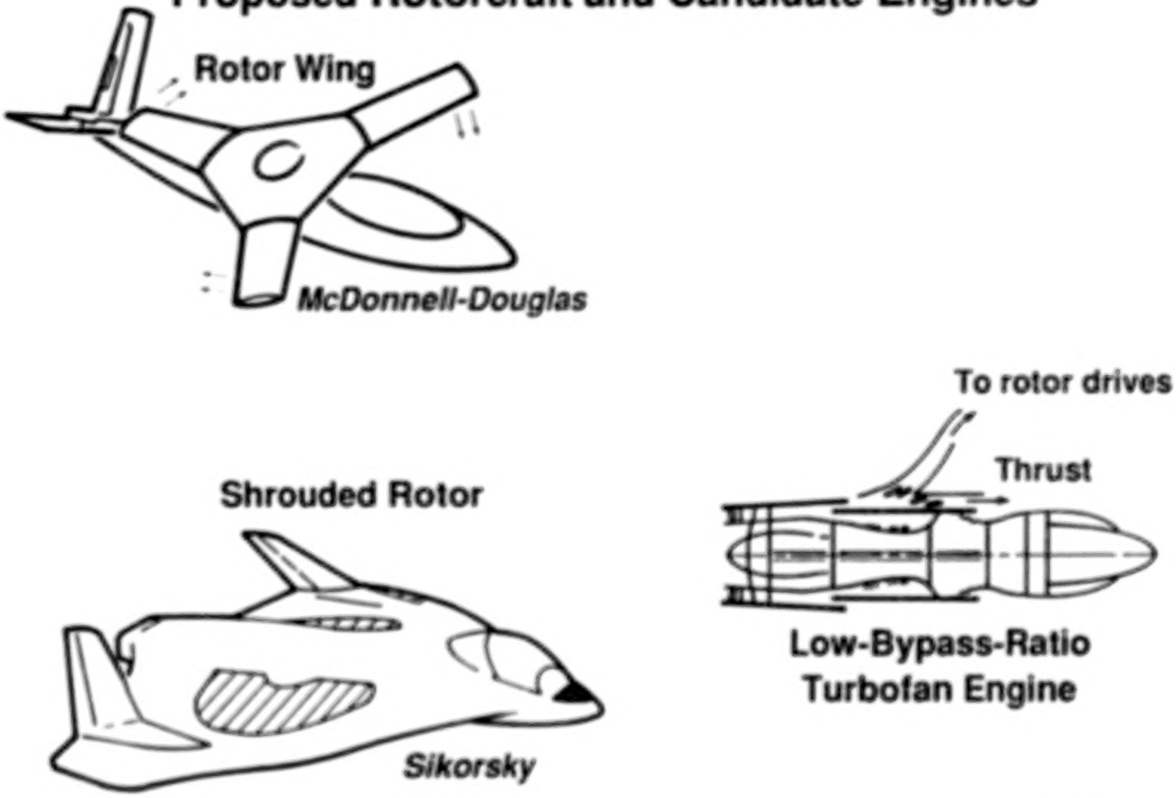
CD-91-54053

Both Bell and Boeing have proposed a fold tilt rotor. Here the rotors are stopped and folded during cruise and the turboshaft engines required for takeoff, landing, and hover become turbofans. This configuration is capable of speeds higher than 450 kn.

The propulsion system that changes from turboshaft to turbofan is called a convertible engine. The example shown here has a fixed-pitch fan that would be decoupled during turboshaft operation. For cruise, a torque converter would be used to bring the fan up to speed. Then, the fan would be locked to the shaft and the torque converter would be drained to prevent power losses and excessive heat generation. A clutch would decouple the rotor drive system.

There are other convertible engine concepts which will be shown later.

Proposed Rotorcraft and Candidate Engines


Rotor Wing
McDonnell-Douglas
Shrouded Rotor
Sikorsky
To rotor drives
Thrust
Low-Bypass-Ratio Turbofan Engine

CD-91-54054

The McDonnell-Douglas rotor wing concept is propelled by a tip-blown rotor at low-speed and hover conditions. For high-speed cruise, the rotor is stopped to become a wing. This configuration is envisioned as capable of speeds higher than 450 kn, and since there is no need for a transmission, the weight penalty is considerably reduced.

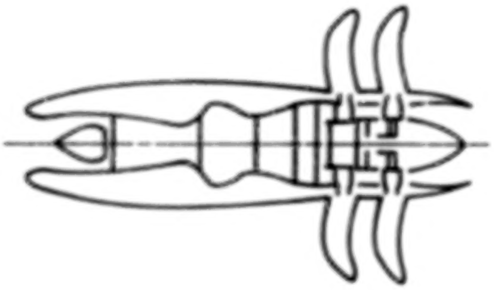
Sikorsky's shrouded rotor concept uses a separate turbine to operate the rotor. Essentially it is a fan-in-wing aircraft.

The minimum weight propulsion system that satisfies the propulsion needs for these aircraft is a low-bypass-ratio turbofan. Such an engine would have high fuel consumption in cruise. However, little, if any, rotorcraft-specific technology appears necessary.

It is possible that some type of dual-mode system with low bypass during rotor operation and high bypass during cruise would be beneficial. Only very detailed vehicle and mission-specific tradeoff studies would determine that.

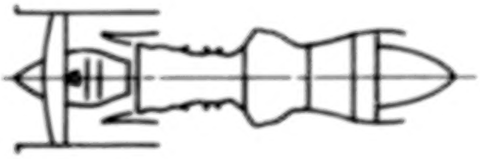
Preferred Convertible Engines

General Electric



Geared Pusher Propfan (UDF)
- **Variable pitch**
- **8 blades in 2 rows**

Allison



Clutched Fan
- **With torque converter**
- **1.6 fan pressure ratio**
- **6.0 bypass ratio**

CD-91-54855

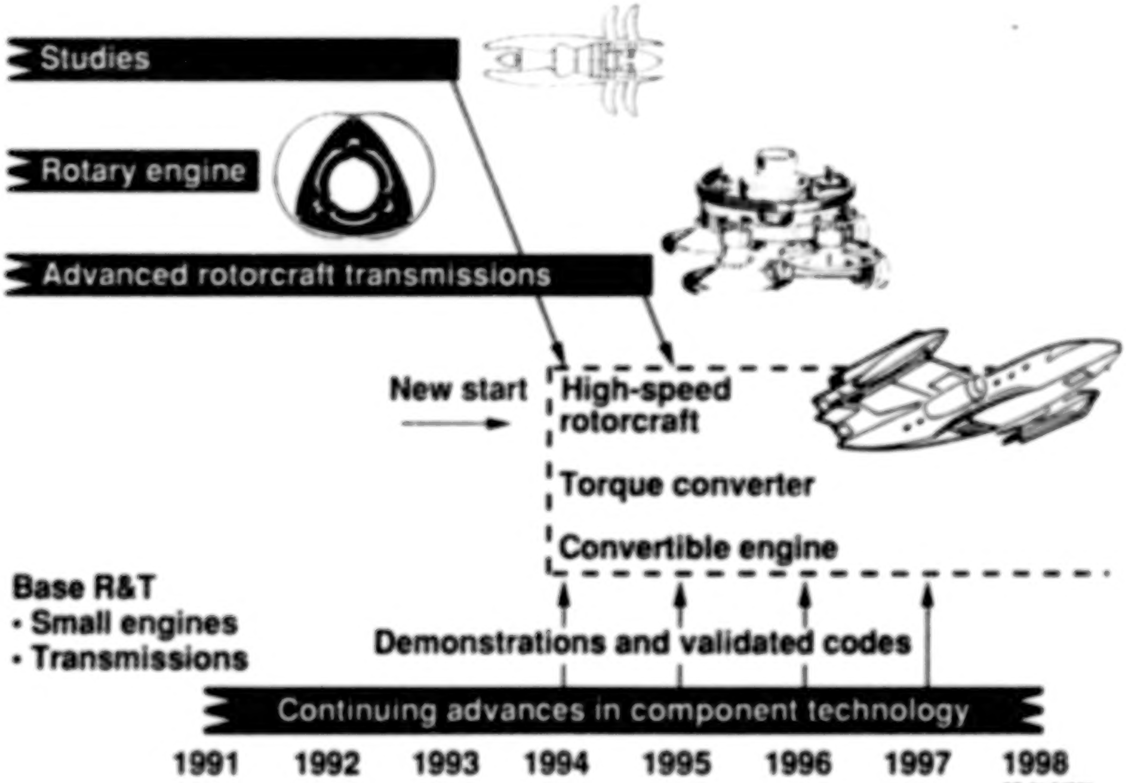
In these propulsion studies, convertible engines have been the first concepts to be fully defined. A number of concepts for convertible engines are considered viable candidates. All of the concepts provide for conversion between turboshaft or turbofan operation.

General Electric suggested a geared unducted fan (UDF) as its first choice, a clutched fan engine using the torque converter as its second choice, and a variable inlet guide vane engine as its third choice.

Allison chose a clutched fan engine using the torque converter concept as its first choice, a turbofan with a variable pitch fan as its second choice, and a variable inlet guide vane engine as its third choice.

The studies have identified additional technology needs. The unducted fan engine requires a gearbox in the hot section - something that has not been done before. And the clutched fan engine requires a torque converter with a power and torque capability 10 times that of current torque converters. Use of variable inlet guide vanes to block fan flow during shaft operation would produce very high power losses during shaft operation unless the technology improves. Use of variable-pitch fans in this application could have weight and aerodynamic penalties unless new technology is applied. Of course all engines will benefit from the application of IHPTET GEN 6 technology.

Rotorcraft and General Aviation Propulsion Program



Studies

Rotary engine

Advanced rotorcraft transmissions

New start

High-speed rotorcraft

Torque converter

Convertible engine

Base R&T
- **Small engines**
- **Transmissions**

Demonstrations and validated codes

Continuing advances in component technology

1991 1992 1993 1994 1995 1996 1997 1998

This figure shows the current activity and the proposed future plan for rotorcraft and general aviation.

The studies that are nearing completion have brought forward some very exciting concepts for airframes and propulsion systems. The rotary engine project has advanced the technology for future production of efficient, high-power density, multifuel engines for commuter aviation. The Advanced Rotorcraft Transmission (ART) project is providing demonstrated, viable transmissions for the Army's next generation of transport and attack-scout rotorcraft. The base research and technology effort in components for small engines and transmissions will continue to infuse into propulsion systems the latest technology for advanced components and validated computer codes, making use of the unique experimental and computational facilities that exist at NASA Lewis.

Looking ahead, a new technology initiative is being prepared for high-speed rotorcraft propulsion - including torque converters, convertible engines, and the related propulsion technologies. We envision that such work, beginning by fiscal year 1994, will be part of a larger, three-NASA-center, high-speed rotorcraft program, to be performed by the Ames, Langley, and Lewis Research Centers, with Lewis concentrating on the propulsion aspects. We project that such a program could enable a 450-kn rotorcraft in the early part of the 21st century.

SUMMARY

This paper has reviewed NASA's propulsion research program for rotorcraft and general aviation, whose expressed strategic goal is to provide the innovative technologies to maintain U.S. rotorcraft and general aviation leadership in the world market and in military capability.

The plan for achieving this goal is (1) to reduce fuel consumption of small engines by 30 percent through the use of advanced cycles, including higher pressure and temperature operation, and the use of improved turbomachinery components technology and ceramic materials; (2) to increase fuel savings and reliability by advanced technology for transmissions; (3) to improve aircraft safety by providing advanced anti-icing and deicing technology; and (4) to achieve high-speed rotorcraft capability through advanced propulsion systems.

NASA's high-speed propulsion studies are aimed at enabling 450-kn rotorcraft. These studies have concluded that no single airframe configuration is best for the three missions of civilian transport, military transport, or military attack. Demanding propulsion technology is required for the configurations that have been identified. If a new technology initiative can begin as early as 1994 for high-speed rotorcraft, including the related propulsion technology, such work could enable a 450-kn rotorcraft early in the next century. The new initiative would provide for work on convertible engines, dual-mode engines, torque converters, gearboxes for hot operation, two-speed transmissions, optimized variable inlet guide vanes and variable pitch fans, and variable-diameter rotors.

The remaining papers in this session of the conference review the current progress and future direction of work in turbomachinery technology for small turbine engines (including compressors, combustors, and turbines), rotary engines, and transmissions. The papers highlight the strong cooperation on propulsion research between NASA Lewis and the U.S. Army, which has existed for two decades.

N91-20113

TURBOMACHINERY AND COMBUSTOR TECHNOLOGY FOR SMALL ENGINES

Lawrence J. Bober and Richard W. Niedzwiecki
NASA Lewis Research Center
Cleveland, Ohio

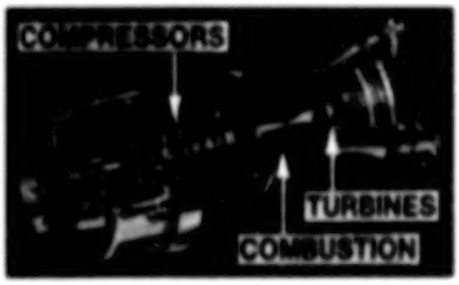
The goal of the Small Turbine Engine Technology Program is to significantly increase thermal efficiency, reliability, and durability of future small gas turbine engines. Significant fuel savings can be achieved through component and cycle improvements as well as through the use of regeneration and uncooled ceramic materials. Recent efforts to identify new regeneration concepts have not been successful, and as a result, no active regenerator research is in progress. Development of uncooled ceramic technology is taking place at the NASA Lewis Research Center under the Department of Energy funded Advanced Turbine Technology Applications Project (ATTAP). Component research includes work on the compressor, combustor, and turbine and will be emphasized in this presentation.

Significantly increase thermal efficiency, reliability, and durability of future small gas turbine engines through

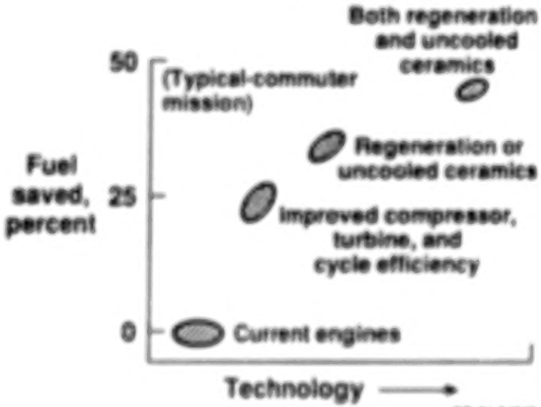
- Improved understandings of flow physics
- Verification and application of advanced codes
- Identification and evaluation of advanced concepts for improved performance

Research areas

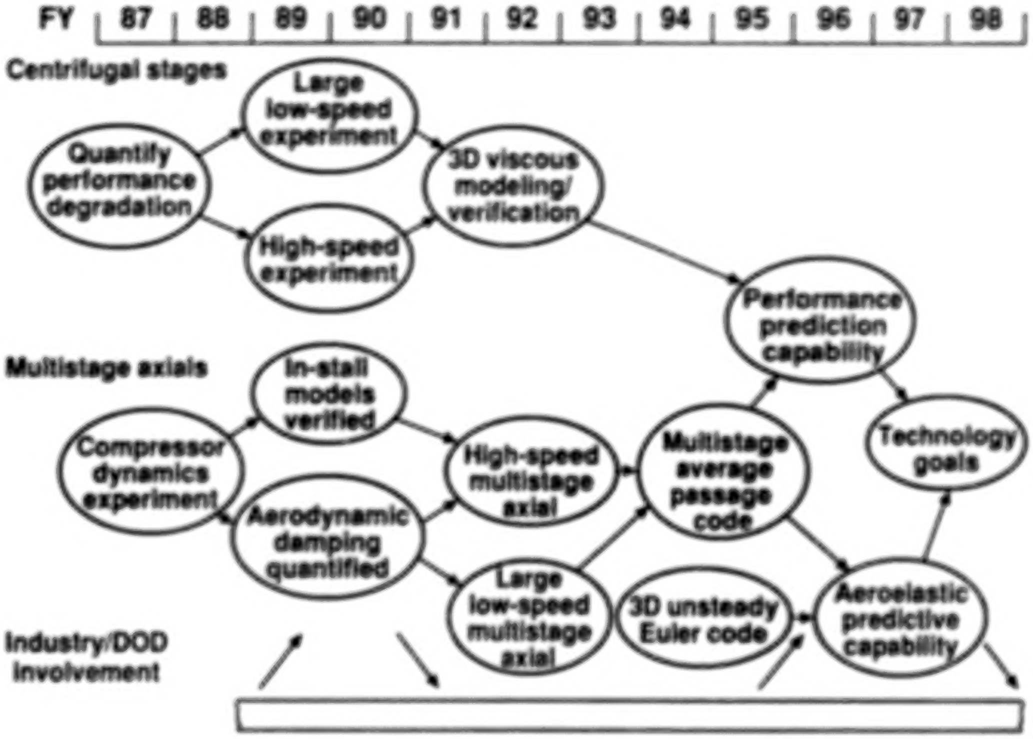
System studies



Technology benefits



Small Compressor Research Program

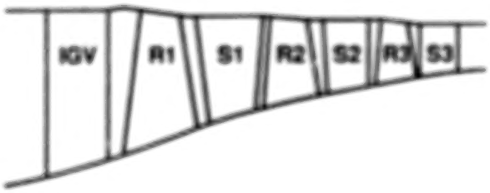

FY
87
88
89
90
91
92
93
94
95
96
97
98
Centrifugal stages
Quantify performance degradation
Large low-speed experiment
High-speed experiment
3D viscous modeling/ verification
Performance prediction capability
Multistage axials
In-stall models verified
Compressor dynamics experiment
Aerodynamic damping quantified
High-speed multistage axial
Large low-speed multistage axial
Multistage average passage code
3D unsteady Euler code
Aeroelastic predictive capability
Technology goals
Industry/DOD involvement

The small compressor research program is a multiyear program coordinated with industry and the military. Results from experimental programs are used to validate computational fluid dynamic codes to provide improved predictive capabilities for both centrifugal and axial compressors. The status of several elements of this program will be discussed in this presentation. The status of the large low-speed experiment will be discussed in another presentation.

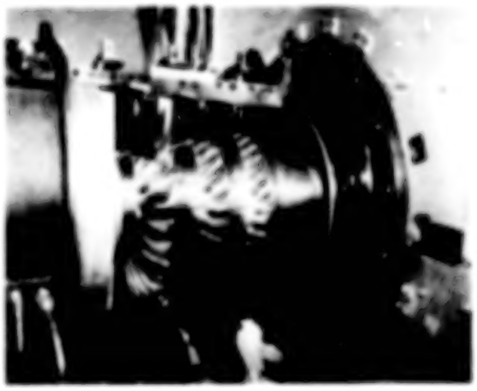
Compressor Dynamics Experiment

Objective

- Measure unstalled performance
- Measure stresses/determine aerodynamic and structural plus mechanical damping
- Measure post-stall performance
- Verify codes



Flowpath



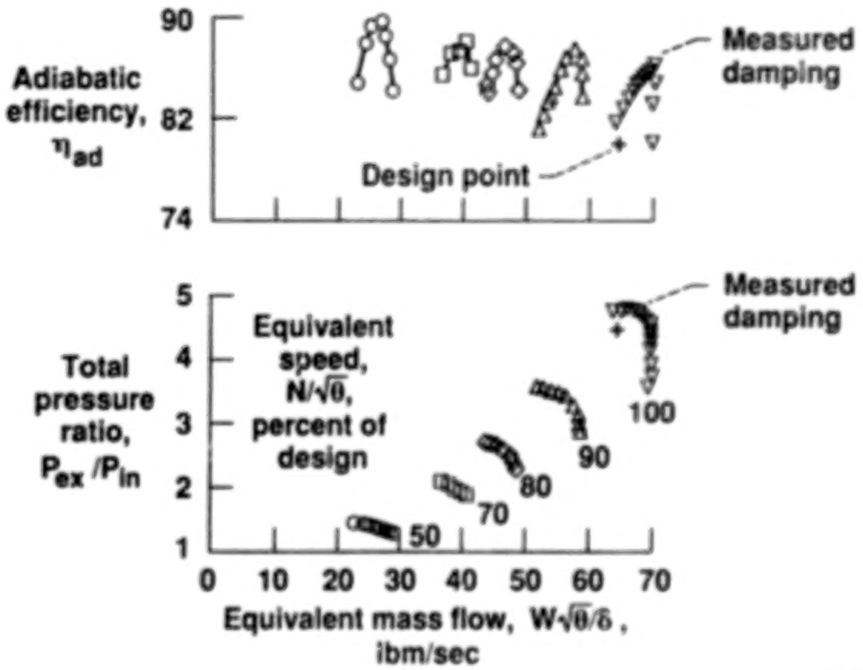
CD-81-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

A three-stage axial compressor was tested over a range of operating conditions to determine unstalled performance, poststall performance, and aerodynamic and structural/mechanical damping of the blades. The compressor first-stage rotor has a diameter of 20 in. and a design tip speed of 1400 ft/sec. The overall pressure ratio was 4.5 at a corrected weight flow of 65.5 lb/sec.

Compressor Dynamics Experiment

Unstalled performance

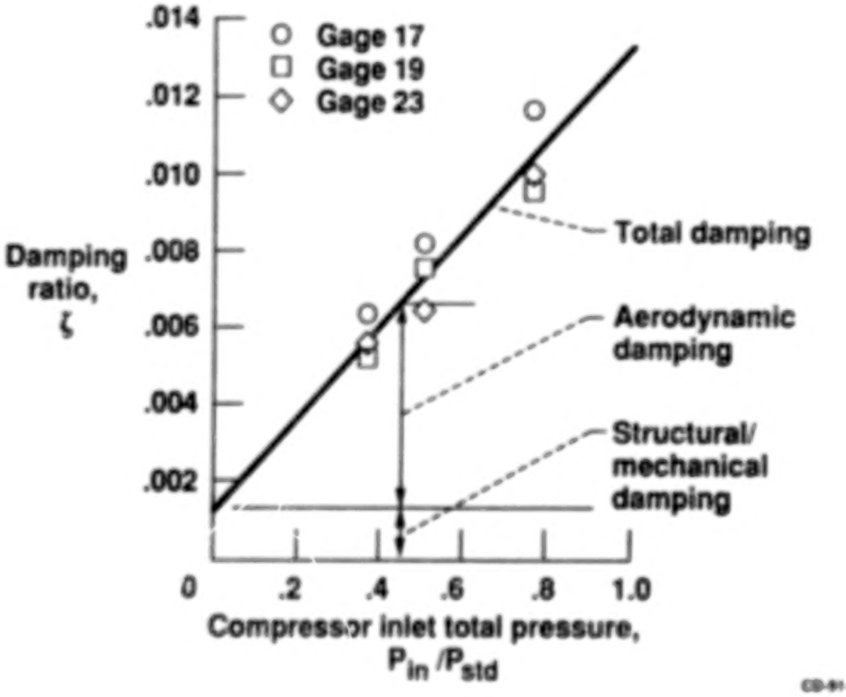


CD-91-54652

Total pressure and efficiency maps of the unstalled compressor are shown here. The design point is indicated with a "plus," and the conditions where the damping of the third-stage rotor was measured is also indicated.

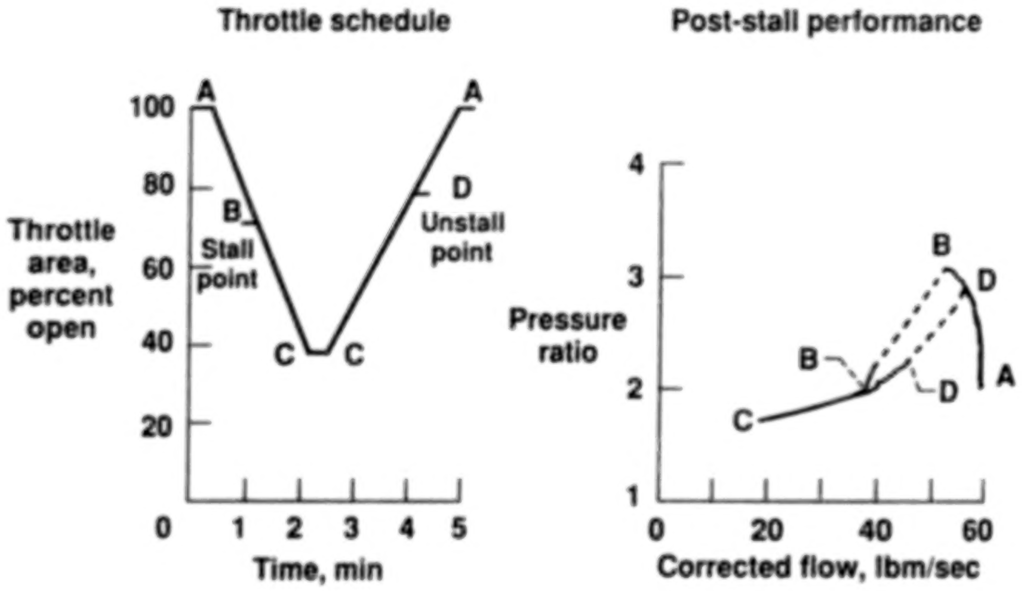
Compressor Dynamics Experiment

Rotor 3 damping first bending mode



The measured damping of the rotor is shown here. Damping values determined from three strain gages are shown. The damping values were determined from the strain gage output together with a one degree-of-freedom structural model. The data points and the line through them correspond to the total damping of the system, consisting of the aerodynamic damping and the structural/mechanical damping. Operating the compressor over a range of inlet pressures resulted in changing aerodynamic damping while the structural damping remained fixed. At the limit of zero inlet pressure, the aerodynamic damping goes to zero and the structural/mechanical damping is equal to the total measured damping.

Compressor Dynamics Experiment



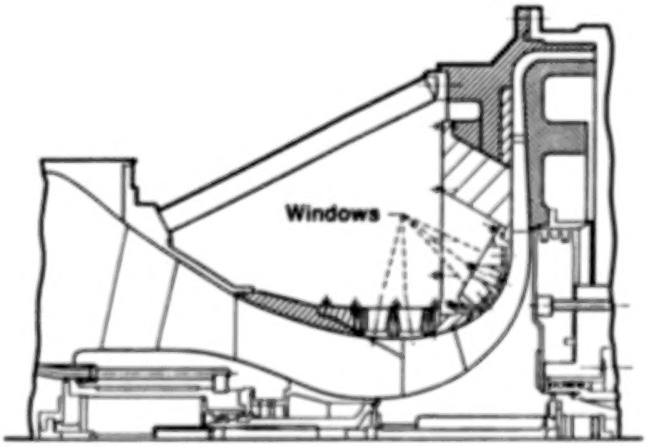
CD-91-54654

The in-stall performance of the three-stage compressor was determined during a stall induced by downstream throttling of the compressor. The throttle schedule used is shown on the left side of the figure. The throttle position was initially fully open as indicated at time zero (point A). The throttle was then gradually closed, passing through the stall (point B), and proceeding well into the stalled region (point C). The throttle was then gradually opened with the compressor unstalling (point D), finally returning to the original operating condition (point A at the right end of the curve). Compressor speed was maintained constant during this process. The resulting compressor performance in terms of pressure ratio is shown on the left side of the figure. At compressor stall there is a sudden drop in pressure ratio and mass flow (from the upper point A to the lower point A), and at compressor unstall there is a sudden rise in pressure ratio and mass flow (from the lower point D to the upper point D). This data is being used to evaluate poststall performance prediction models.

4:1 Pressure Ratio Compressor

Objective:

Obtain aerodynamic performance and laser anemometer measurements of the flow fields within a high-speed centrifugal compressor.



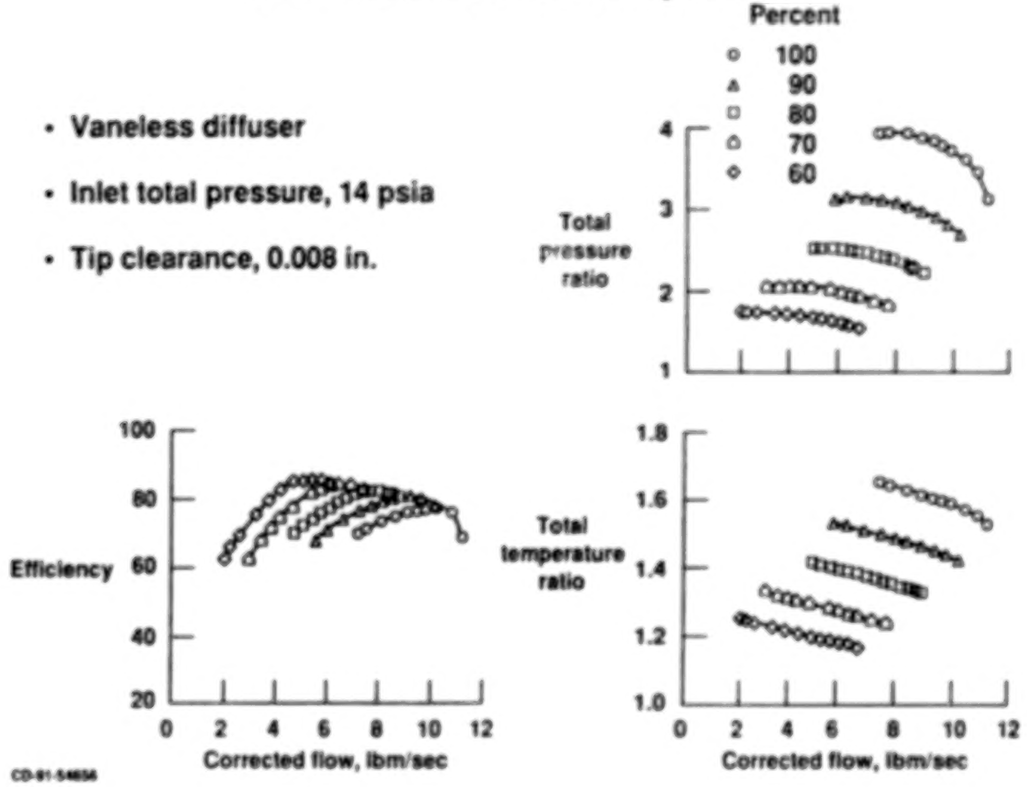
CD-91-54655

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

A small centrifugal compressor is being tested to provide data that will allow an improved understanding of the flow fields in centrifugal compressors. The compressor is a scaled-up version of an Allison design. At the design point corrected tip speed of 1600 ft/sec and corrected weight flow of 10 lb/sec, the compressor had a pressure ratio of 4:1. This compressor incorporates splitters in the rotor blade passages. In addition to measuring overall performance, detailed flow-field measurements will be made in the rotor blade passages. A photograph of the compressor and a cutaway drawing of the hardware are shown here. Areas indicated by arrows are casing windows which allow optical access to the rotor blade passages for laser velocimetry measurements.

4:1 Pressure Ratio Compressor

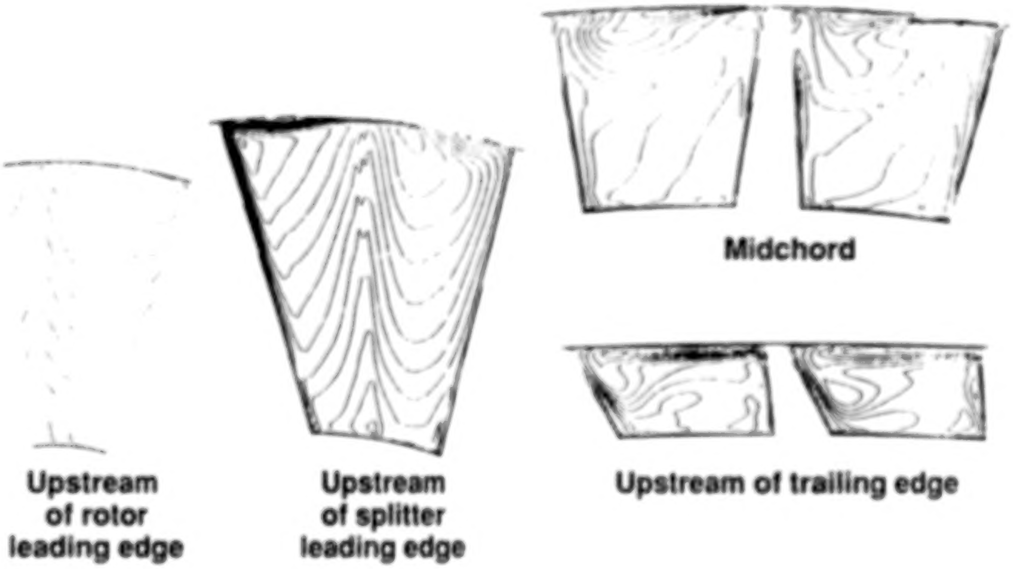
- Vaneless diffuser
- Inlet total pressure, 14 psia
- Tip clearance, 0.008 in.



At this time, the compressor has been tested with a vaneless diffuser. Experimental results for this configuration at atmospheric inlet pressure and design tip clearance are shown in the figure. Pressure ratio, temperature ratio, and efficiency maps are included. Tests have been conducted at several inlet pressures from 3 psia to ambient and also with several spacings between rotor and casing ranging from design to 5 times the design value. The compressor will be tested next with a vaned diffuser followed by laser velocimetry measurements in the rotor blade passages. There are currently no detailed velocity measurements for a centrifugal compressor with splitters. Thus, it is very important to obtain laser velocimetry data to allow validation of the three-dimensional, viscous CFD codes that are being used to predict the flow in this type of compressor.

4:1 Pressure Ratio Compressor

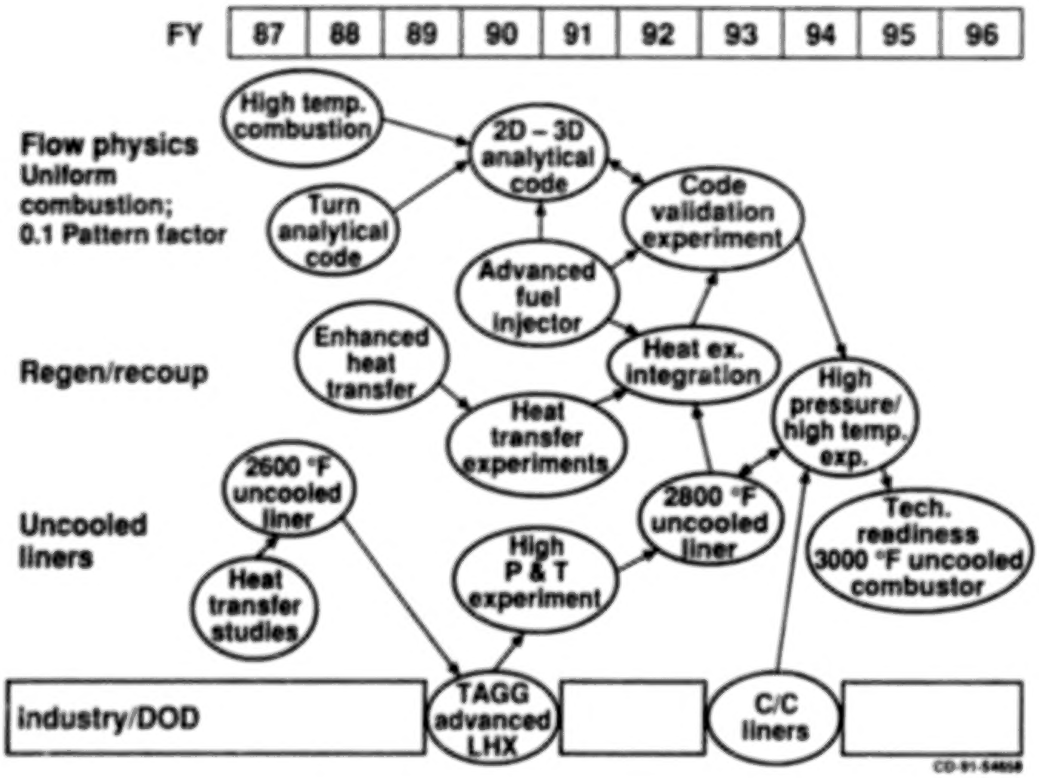
Predicted Mach Number Contours



CD-91-54657

Three-dimensional, viscous calculations of the flow field in the 4:1 pressure ratio compressor have been performed using the turbomachinery code developed by Dawes. Tip clearance for both the blade and splitter are included in the calculations. Flowfield predictions are shown in this figure for a flow rate near the design value. Mach number contours are shown at several planes approximately perpendicular to the flow path. Results are shown slightly upstream of the rotor blade leading edge, just upstream of the splitter leading edge, about midway between the splitter leading and trailing edges, and slightly upstream of the blade and splitter trailing edges. For each contour plot, the hub surface is at the bottom, the shroud surface is at the top, the blade suction surface is at the left, and the blade pressure surface is at the right. The splitter is in the center of the passage. The results indicate that the flow upstream of the splitter exhibits the expected higher velocities near the blade suction surface compared to the pressure surface and a slightly reduced velocity at midpassage ahead of the splitter leading edge. Just ahead of the blade and splitter trailing edges, the gradients present ahead of the splitter leading edge have disappeared, and the flow in both passages is similar with only minor differences present.

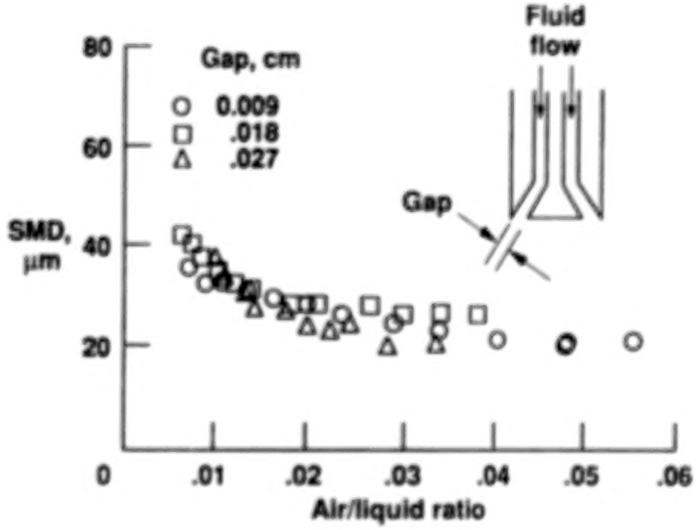
Reacting Flow Research

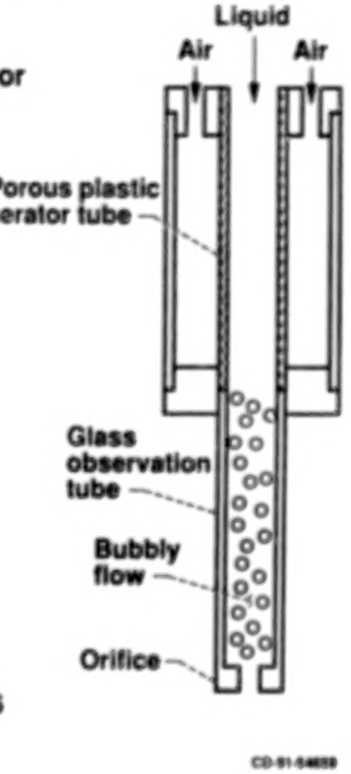

FY
87
88
89
90
91
92
93
94
95
96
Flow physics
Uniform combustion; 0.1 Pattern factor
Regen/recoup
Uncooled liners
High temp. combustion
Turn analytical code
2D – 3D analytical code
Code validation experiment
Advanced fuel injector
Enhanced heat transfer
Heat transfer experiments
Heat ex. integration
High pressure/ high temp. exp.
2600 °F uncooled liner
Heat transfer studies
High P & T experiment
2800 °F uncooled liner
Tech. readiness 3000 °F uncooled combustor
Industry/DOD
TAGG advanced LHX
C/C liners
CD-91-54658

The overall program in combustor research for small turbine engines is summarized in this figure. Research on the flow physics of reacting flows emphasizes analytical and experimental studies aimed at providing very uniform flow fields at the combustor exit. Research on uncooled liner technology is in progress to allow operation at 3000 °F. Research efforts are coordinated with industry and the military. Combustion research on regenerators and recuperators has been delayed because efforts to identify attractive new concepts for these components have not been successful. However, new ideas, which would allow more efficient or lighter weight regenerators and recuperators, are still being sought. Research in progress in several areas will be summarized in this presentation.

Effervescent Fuel Injection

Objective:
Evolve advanced fuel atomization techniques for small gas turbine combustors.

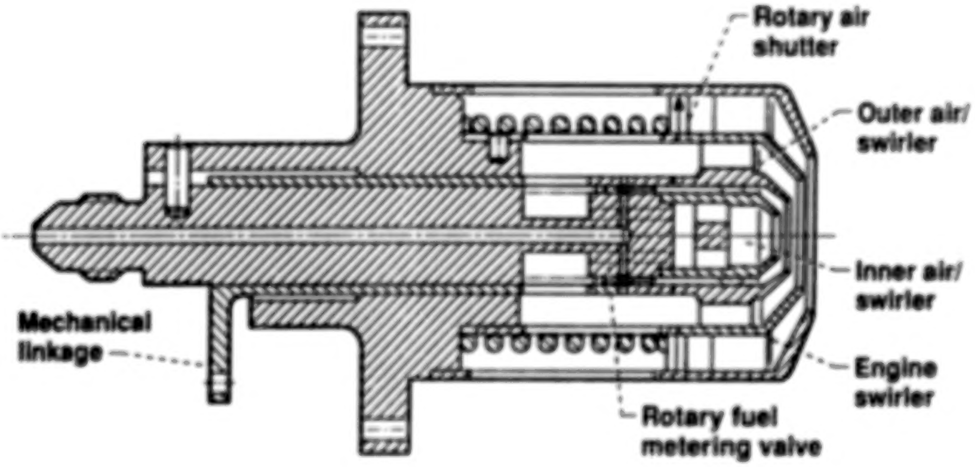



Liquid
Air
Air
Porous plastic aerator tube
Glass observation tube
Bubbly flow
Orifice
CD-91-54659

An experimental program is in progress to develop advanced fuel injectors with improved fuel atomization characteristics. The approach shown conceptually at the right involves mixing air with the fuel prior to injection in the combustor. This mixing is accomplished by using a porous aerator tube. Results for an experimental injector tip configuration shown at the left indicate that small droplet sizes can be obtained for a wide range of injected air to liquid fuel ratio and that the droplet size is not sensitive to the size of the injector flow passages. In addition, small droplet sizes are available with fairly large passages, an advantage for small engines.

Advanced Fuel Injector

Objective:
Evolve advanced fuel injection technology for future small gas turbine applications capable of doubling the operating range of current fuel injectors.



Variable geometry pure airblast

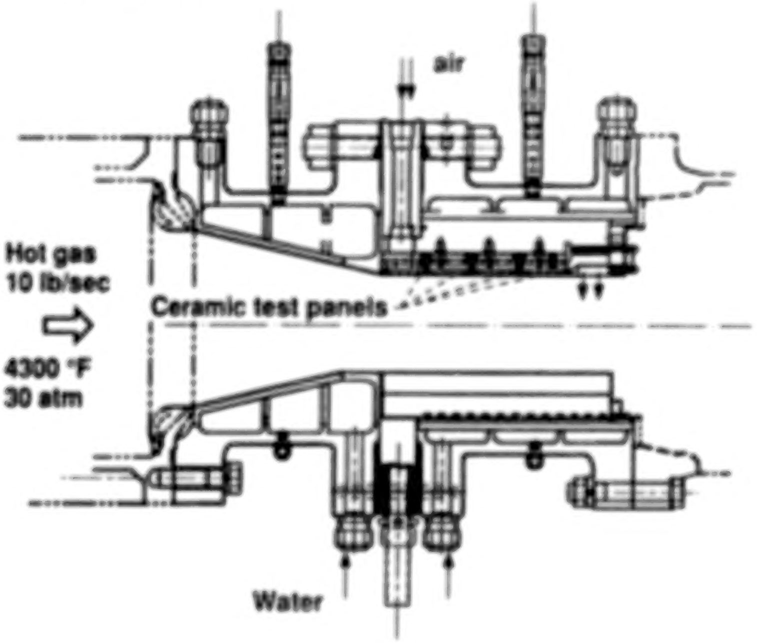
CD-91-[illegible]

Future aircraft missions will require combustors to provide considerably higher temperature rises. This requirement imposes severe problems for fuel injectors that will be required to operate over wider operating ranges. A contractual program was formulated to meet this need by evolving injectors that double the operating range. An example of one of the injectors being developed in this joint NASA/Army/Allison program is shown in this figure. The injector is an air blast type. Kinetic energy of the combustor airflow is used to break up the fuel into small, uniform droplets conducive to uniform combustion. Air blast injectors are low fuel pressure loss devices in contrast to conventional pressure atomizers that are high fuel pressure drop devices. One of the major advantages of air blast atomizers is that small restrictions, prone to clogging, are eliminated.

Ceramic Matrix Liner Test Rig

Objective:

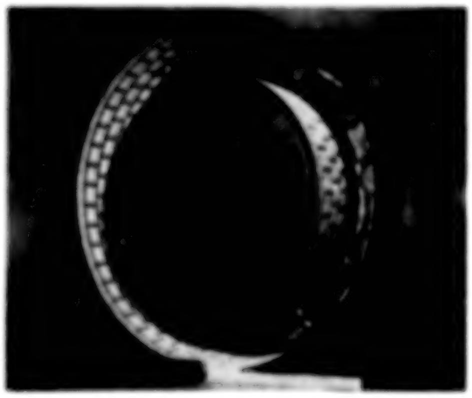
Evaluate advanced ceramic/composite materials under advanced engine conditions.

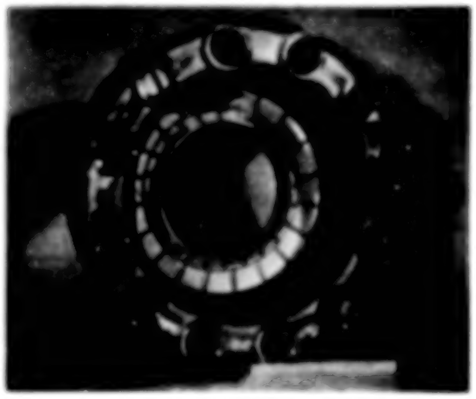


A new test rig for evaluation of ceramic/composite combustor liner materials under realistic engine conditions has been developed at the NASA Lewis Research Center. The test rig can evaluate up to 16 samples simultaneously. The samples are placed in four quadrants, each of which has independently controlled coolant flow rates. The samples are exposed to a hot gas source at high pressure to simulate combustor operating conditions. This approach allows the controlled testing of materials, apart from the issue of combustor design and characteristics, such as gradients in combustion gas temperature and composition. This test rig facility provides an excellent opportunity for joint programs between NASA and engine or ceramic/composite manufacturers for evaluating new combustor materials.

Ceramic Matrix Combustor

- Advanced cycle 3000 °F exit temperature
- Compact reverse flow annular design
- State-of-the-art design methodology
  - 3D internal reacting aerodynamics
  - 2D wall temperature distribution





Payoffs

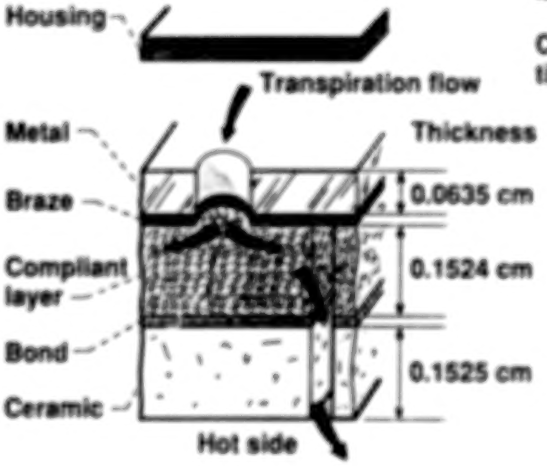
- 50% reduction in cooling air requirement compared to Lamilloy or effusion
- Low emissions and smoke
- Improved efficiency, pattern factor, and durability

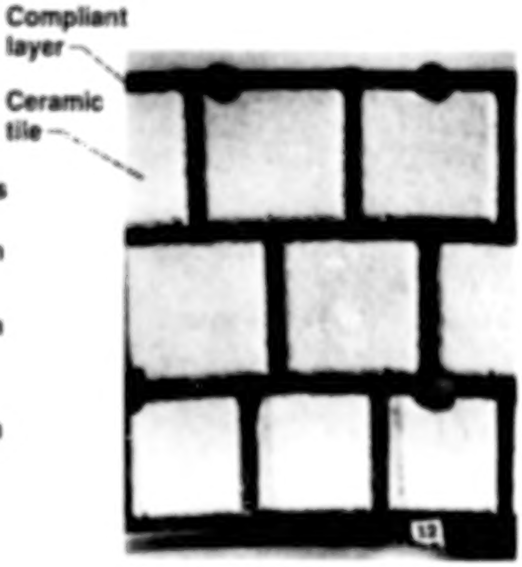
CD-91-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

An advanced liner concept, the ceramic matrix liner, was applied to a combustor as shown in the figure. The combustor is a compact, reverse flow, annular combustor designed with a three-dimensional reacting flow computer program and a two-dimensional heat transfer analysis. This advanced ceramic matrix combustor has the potential of reducing combustor cooling air by 50 percent compared to Lamilloy or effusion cooled combustors. In addition, the combustor promises low emissions and smoke as well as improved efficiency, pattern factor, and durability. The ceramic material is visible in the photographs as small light colored rectangular features on the interior surfaces of the combustor. This research, performed under a joint NASA/Army/Allison program, has applicability to future DOD engines.

Ceramic Matrix Combustor


Housing
Transpiration flow
Metal
Thickness
Braze
0.0635 cm
Compliant layer
0.1524 cm
Bond
0.1525 cm
Ceramic
Hot side


Compliant layer
Ceramic tile

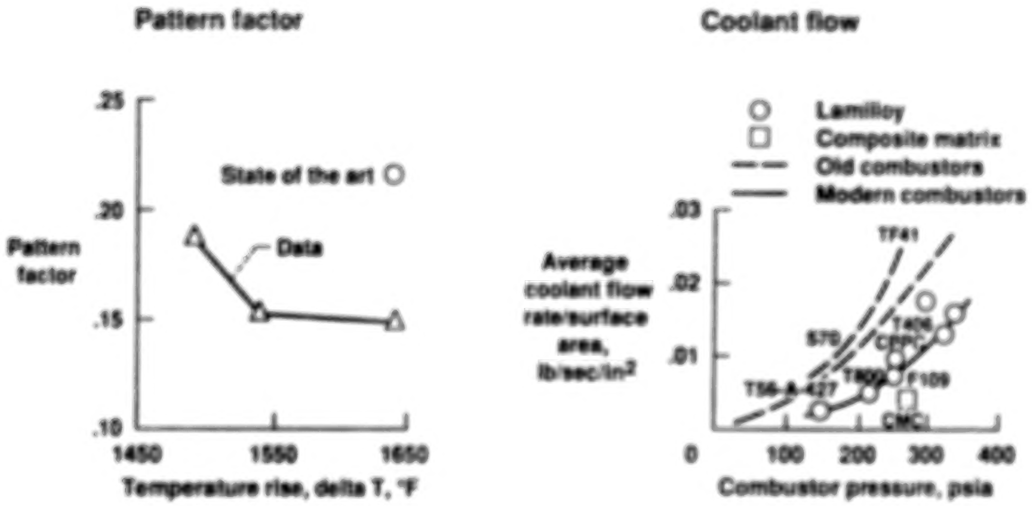
CD-91-[illegible]

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

A cross section of the ceramic matrix combustor liner is shown here on the left. Structural loads in the combustor are carried in a metallic substrate. The ceramic liner material is attached to this metallic substrate with a porous, compliant pad that allows cooling flow to pass between the ceramic tiles and isolates the ceramic tiles from structural loads. The cooling air enters through holes in the metallic substrate, passes through the porous pads, and exits between the individual ceramic tiles into the combustor. On the right, the figure shows a photo of a portion of the combustor interior with the ceramic tiles and coolant passages. The tiles are attached to the compliant layer which can be seen through the passages.

Ceramic Matrix Combustor

Test results

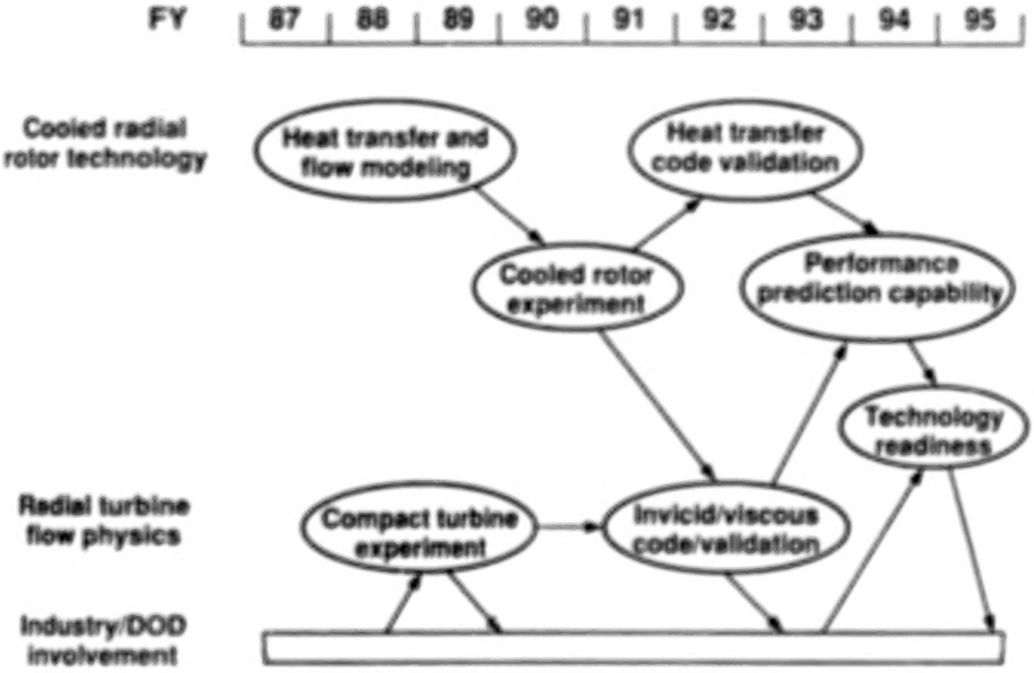

Pattern factor

Coolant flow

CD-91-54884

Test results shown on the right of the figure demonstrate that approximately 50 percent lower coolant flow rates are required for the ceramic matrix combustor liner compared to a modern Lamilloy combustor. Even larger reductions relative to older design combustors are possible. Since less liner cooling flow is required, more airflow is available for other combustor processes. Introducing this "saved" air into the combustion zone results in more uniform burning that produces the reductions in pattern factor shown on the left side of the figure.

Small Turbine Research Program


FY | 87 | 88 | 89 | 90 | 91 | 92 | 93 | 94 | 95

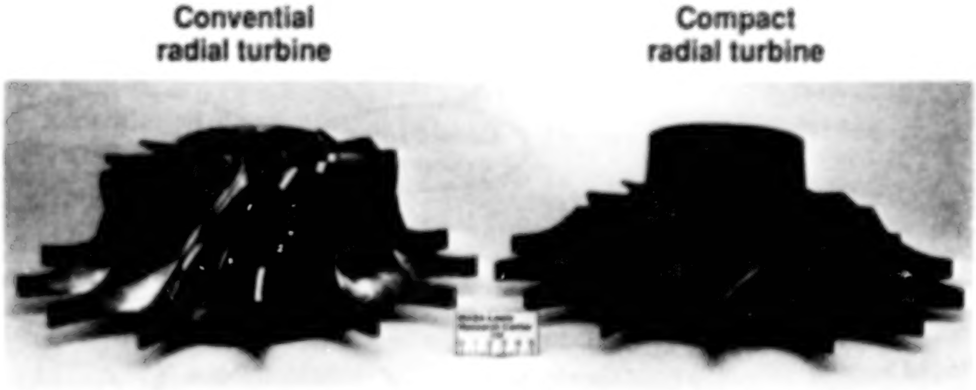
Cooled radial rotor technology
Heat transfer and flow modeling
Heat transfer code validation
Cooled rotor experiment
Performance prediction capability
Technology readiness

Radial turbine flow physics
Compact turbine experiment
Invicid/viscous code/validation

Industry/DOD involvement

The small turbine research program is a multiyear program coordinated with industry and the military. Results from experimental programs are used to validate computational fluid dynamic and heat transfer codes to provide improved predictive capabilities for radial inflow turbines. The status of several elements of this program will be discussed in this presentation. Although the turbine research program has emphasized radial inflow turbines for the last several years, the program is being expanded to include work on axial flow turbines. A transonic linear turbine cascade facility will be described in another session. Plans for turbine testing using rotating rigs are in the formulation process.

Conventional/Compact Radial Turbines



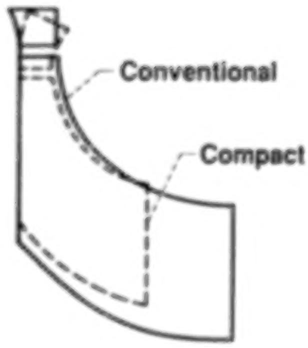
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

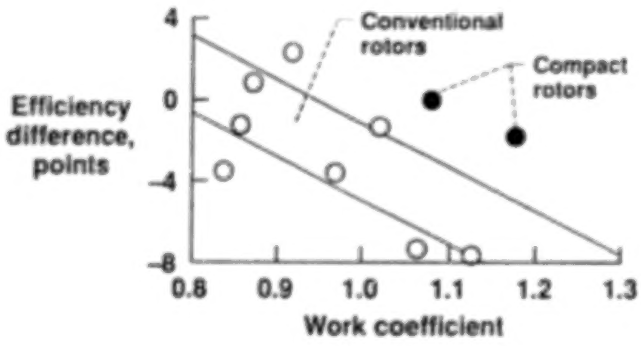
CD-91-[illegible]

A compact radial turbine, 40 percent shorter than conventional radial inflow turbines, was designed and fabricated by Pratt Whitney and tested at the Lewis Research Center. A conventional radial turbine is shown here on the left and the compact radial turbine is shown on the right. The compact radial turbine was built with a 14.4-in. tip diameter and has a design point pressure ratio of 5:1. The reduced length of the compact turbine offers significant weight savings relative to the conventional turbine.

Compact Radial Turbine

Objective:
Conduct fundamental experiments to obtain detailed data for model development and code validation of compact/lightweight radial turbine.

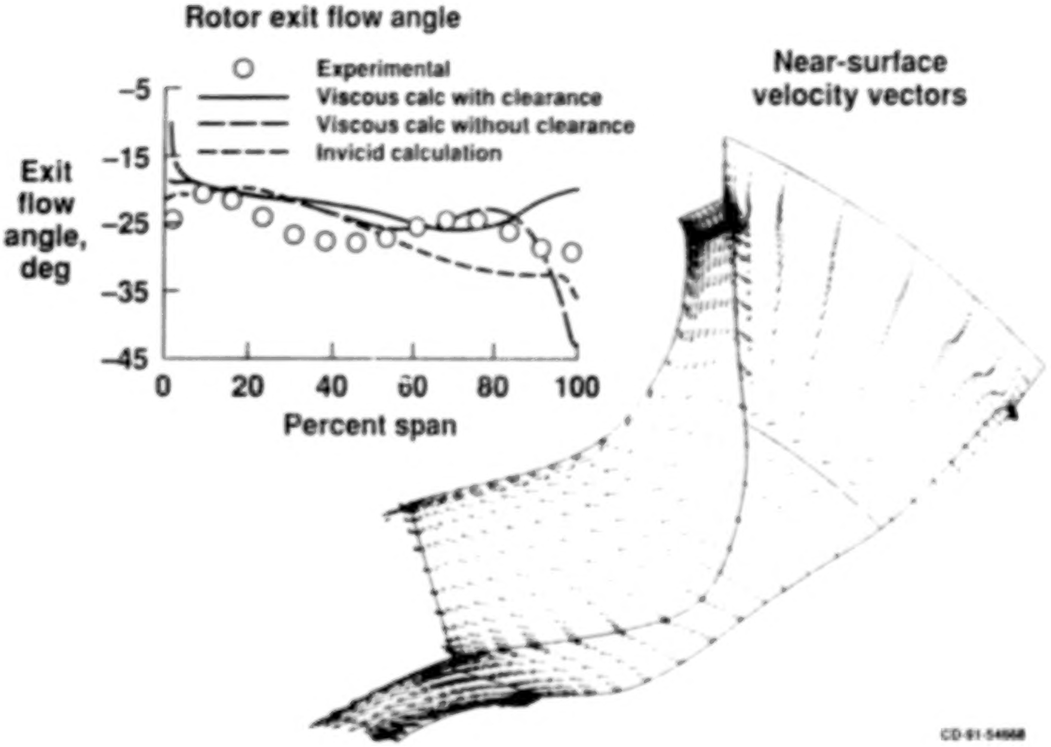




CD-91-[illegible]

Another comparison of the compact and a conventional radial inflow turbines is shown in this figure. Test results for the compact turbine, shown here as symbols, indicate that the efficiency is comparable to, or somewhat higher than, that for a conventional radial inflow turbine over a range of operating conditions. The measured efficiency levels, together with the short length and reduced weight of the compact radial turbine, make this concept a very attractive one for future radial inflow turbine applications.

Compact Radial Turbine Analysis

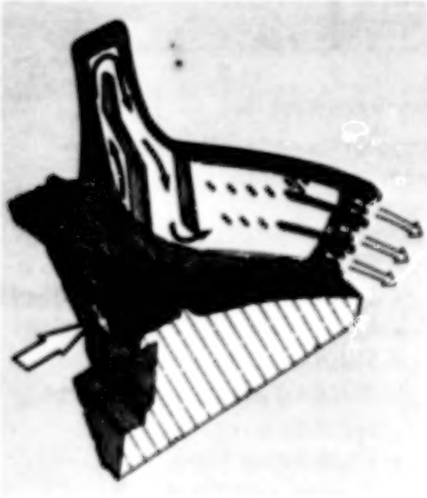


CD-91-54668

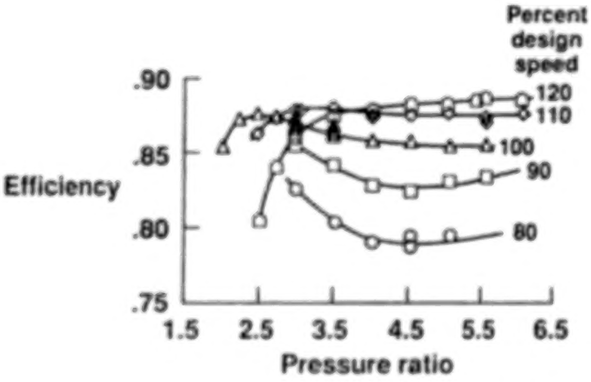
The compact radial turbine has been analyzed using a three-dimensional average passage analysis developed at Lewis by Adamczyk. This code, which was previously used only for axial turbomachinery components, was modified to allow radial inflow geometries and blade tip clearance. Computational results were obtained for various levels of modeling sophistication including inviscid with no clearance, viscous with no clearance, and viscous with clearance. A portion of the calculated results are shown on the left side of the figure where turbine exit flow angle is plotted versus percent span for these calculations. Results indicate the sensitivity of the calculation to viscous effects and blade clearance. Although the viscous flow results with tip clearance are in reasonable agreement with the experimental data, it should be noted that the clearance used in the calculation was not an exact match to the actual tip clearance distribution present in the experiment. Based on the sensitivity of the results to including clearance, it is reasonable to assume that an accurate specification of clearance distribution will be required to accurately predict turbine exit flow-field characteristics. Shown on the right side of the figure are velocity vectors near the rotor and hub surfaces from the viscous calculation with blade clearance. The velocity vectors near the intersection of two surfaces show the effect of the tip clearance flow as a sudden change in flow direction.

Cooled Radial Turbine

Objectives
- Develop rotor cooling technology to increase operating temperature of radial turbines
- Demonstrate the fabrication of a cooled radial turbine
- Obtain data to assess and improve heat transfer and aerodynamic codes



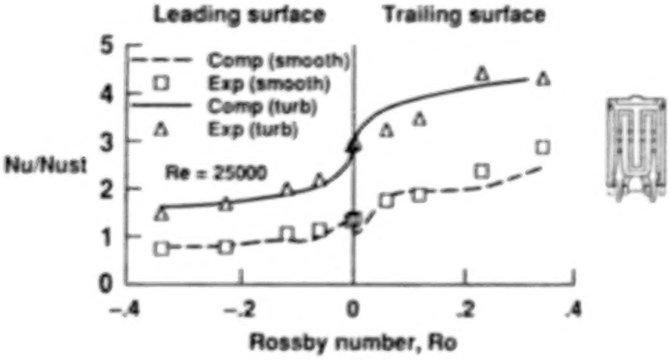
Stage performance of uncooled configuration of cooled radial turbine



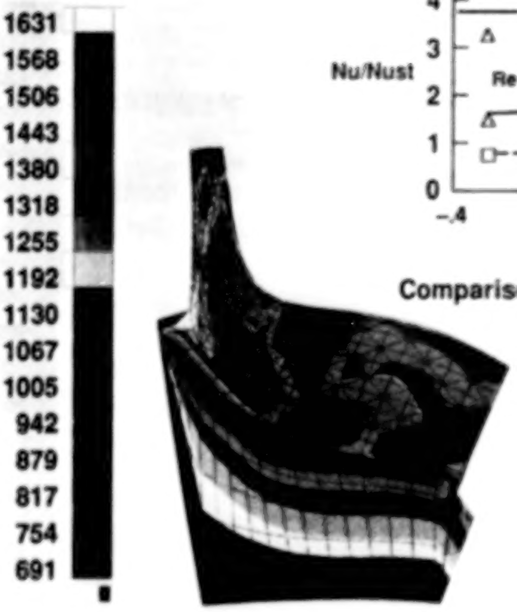
CD-91-54669

Although cooled blades for axial flow turbines are quite common, radial inflow turbines do not currently use cooled blades. The complex fabrication techniques required to build a cooled radial inflow rotor are being addressed by the NASA Lewis Research Center under a contract to the Allison Gas Turbine Division of General Motors. The use of a cooled radial turbine will allow the use of higher turbine inlet temperatures in small engines. Under this program a turbine rotor was fabricated with integrally cast cooling passages. Final machining of the rotor is in progress. In the meantime, a solid rotor was tested to provide aerodynamic performance data. The turbine, built with a 14.4-in. tip diameter, has a design point pressure ratio of 4:1. Turbine efficiency results are shown here. The performance is typical of a modern design radial turbine. The cooled version of the turbine will be tested in a warm turbine test rig in which the ratio of coolant temperature to turbine inlet temperature will be the same as in an actual engine. In addition to measuring the performance of the cooled radial inflow turbine, the program objectives include obtaining detailed data for validation of design and analysis codes for aerodynamics and heat transfer. Additional tests are planned for acquiring rotor surface static pressure information, flow-field data using laser velocimetry, and coolant passage heat transfer data.

Turbine Coolant Passage Heat Transfer Analyses



Comparison with rotating serpentine passage experimental data



Thermal analysis of cooled radial turbine

Analysis features

- Rotational effects
- Entrance and turning effects
- Leading edge impingement
- Ribbed turbulators
- Pin fins (including lateral ejection)
- Flow branching
- Finned passages

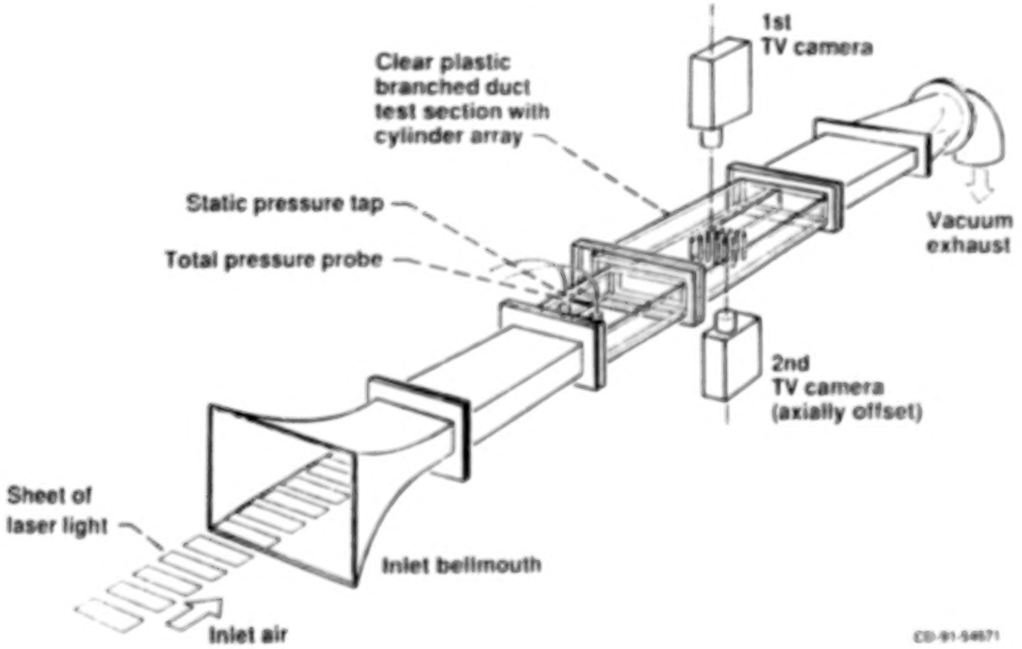
CD-91-54670

To better understand the aerodynamics and heat transfer in turbine blade cooling passages, a one-dimensional analysis for aerodynamics and heat transfer has been developed. This analysis includes many physical and geometric features that are used for turbine blade cooling passages. A list of these features is included on the figure. The analysis has been applied to a rotating serpentine cooling passage for which heat transfer data are available. Analysis results are compared to the experimental data. Excellent agreement between experiment and analysis was obtained for both smooth and rough cooling surfaces. This coolant code was used in making a rotor thermal analysis of the cooled radial inflow rotor described in the previous figure. The predicted temperature distribution on the surface of the blade is shown. Heat transfer results from the cooled turbine test program will be used to further validate the analysis.

Turbine Cooling Passage Experiment

Objective:

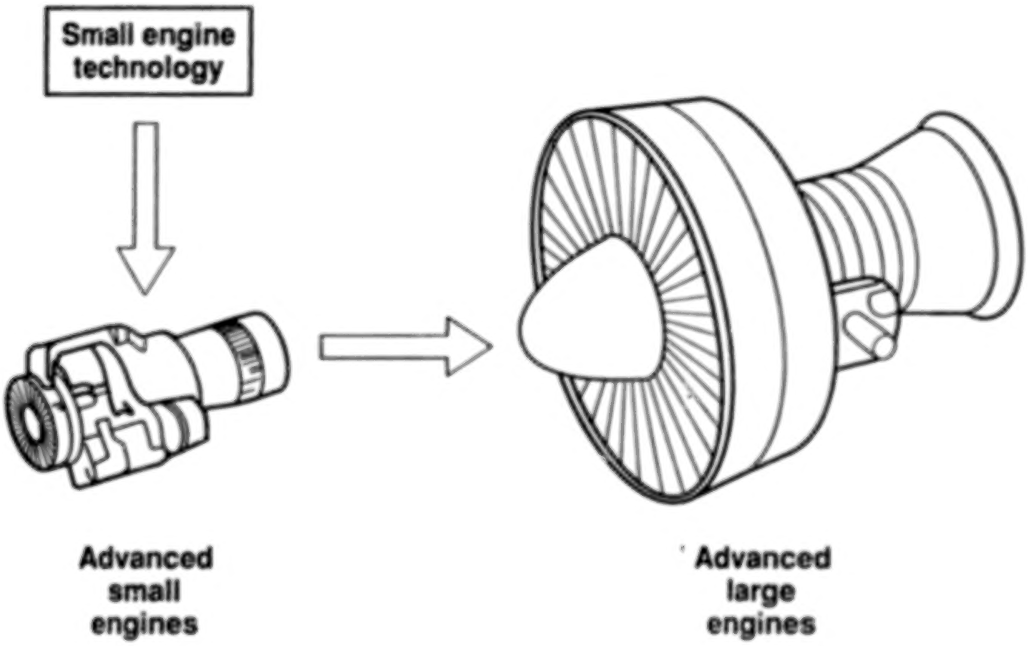
Obtain detailed flow-field and heat transfer measurements in a simulated turbine cooling passage.


1st
TV camera
Clear plastic
branched duct
test section with
cylinder array
Static pressure tap
Total pressure probe
Vacuum
exhaust
2nd
TV camera
(axially offset)
Sheet of
laser light
Inlet bellmouth
Inlet air

CD-91-54671

As coolant passage analyses become more sophisticated, more detailed flow-field and heat transfer measurements will be required to validate the analyses. To provide this kind of detailed data, a relatively large simulated cooling passage is being built to allow measurements in a simplified, but representative, geometry. The experimental apparatus is shown in the figure. The flow passage divides and one branch of the passage contains pins similar to those which would be used in a radial inflow turbine to increase turbulence and heat transfer. In addition to detailed surface static pressure and heat transfer measurements, flow visualization will be performed using laser light sheets.

Small Engine Technology and Large Engines



CD-91-54672

The immediate beneficiary of the NASA small engine program is the small engine manufacturer. However, as manufacturers of engines for large commercial and military aircraft strive for higher efficiency through higher overall pressure ratios, the high pressure compressor blades, the combustor, and the high pressure turbine blades can become quite small. These components can approach the size of those used in the small engines of commuter and general aviation aircraft. Because of this, the NASA small engine program, in addition to advancing technology for small engines, is laying the groundwork for future advances in large engine technology.

CONCLUDING REMARKS

Turbomachinery and combustor research in progress at the NASA Lewis Research Center for small engines has been presented. Steady-state performance for both stalled and unstalled operating conditions were measured for a three-stage axial compressor. In this same experiment aerodynamic and structural/mechanical damping were measured. Performance measurements are in progress for a single-stage centrifugal compressor, and detailed flow-field measurements are planned to provide a better understanding of centrifugal compressor flow fields. Advanced fuel injector designs are being investigated experimentally to provide improved fuel atomization and increased operating range. Ceramic and ceramic composite materials are being investigated in coupon testing and in an actual combustor to allow increased combustor temperature or decreased combustor cooling flow. An advanced, compact radial inflow turbine was tested to demonstrate the ability to reduce turbine length while maintaining performance. An experimental program to demonstrate the feasibility of a cooled radial turbine is in progress. A simulated cooling passage experiment as well as a warm turbine test are planned. State-of-the-art computational fluid mechanics and heat transfer methods are being used in conjunction with the experimental programs to gain a better understanding of the physical processes involved in small engine components. In addition, improved analysis methods will result, which will allow the design of improved components.

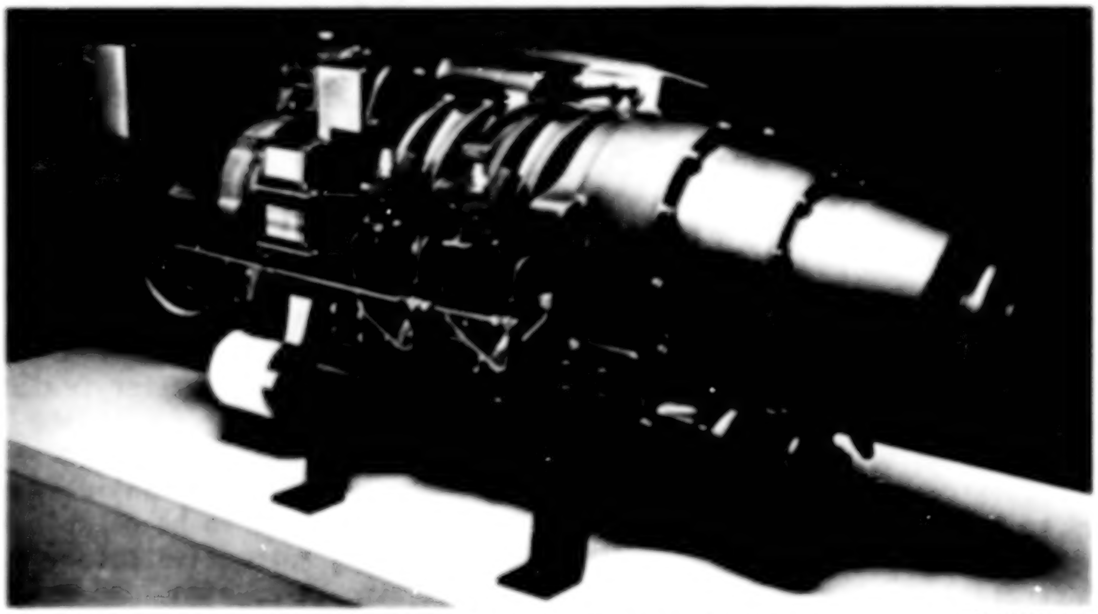
N91-20114

ROTARY ENGINE TECHNOLOGY

Edward A. Willis and John J. McFadden
NASA Lewis Research Center
Cleveland, Ohio

NASA's original goal in this program was to combine some of the better features of reciprocating and gas-turbine engines, thus obtaining a superior powerplant for light aircraft and a host of related uses. Based on a series of engine and aircraft/mission studies conducted in the early 1980's, the rotary (Wankel) engine was perceived to have many of the desired features and was judged to be capable of considerable further development. At the time, existing automotive rotary engines had demonstratably high power outputs in a compact, smooth-running, and reliable package, but were less fuel efficient than comparable reciprocating engines. Accordingly, the basic thrust of the NASA Rotary Engine Technology Enablement Program was (and is) to bring the rotary's efficiency up to the level of a comparably sized diesel, without sacrificing its other desirable features.

Overview of NASA Objectives



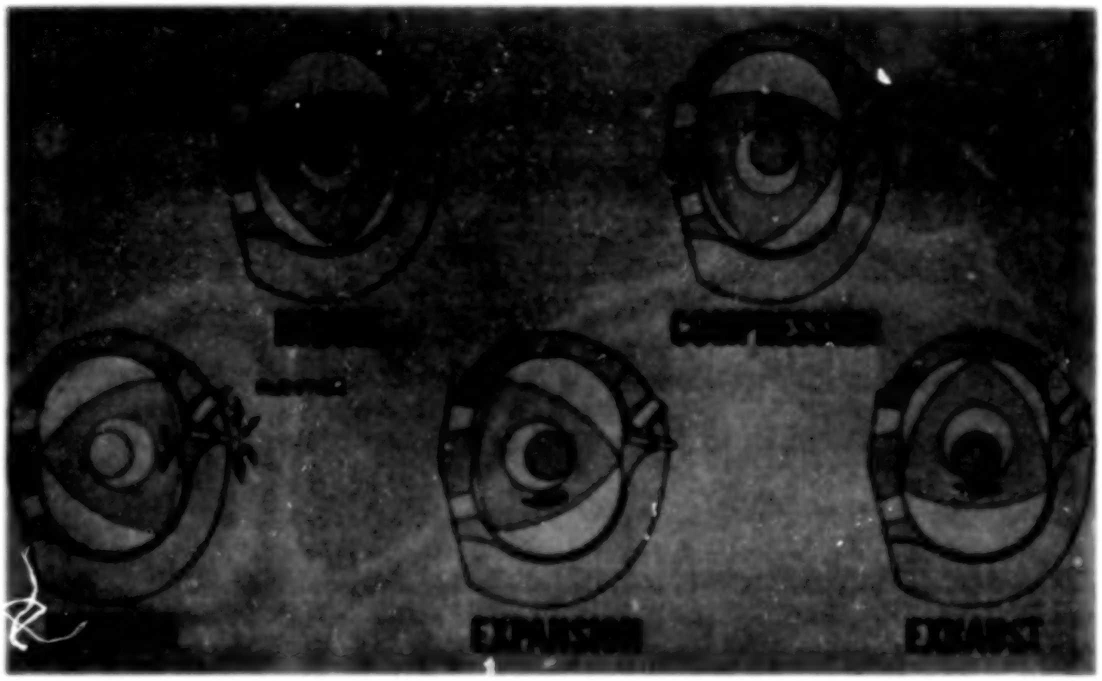
CD-91-53675

- Multifuel capability (gasoline, jet, diesel, and alcohol fuels).
- Cruise BSFC: 0.36 lb/hp-hr, or better.
- Power density (maximum): 5 hp/in.^{3}, or better.
- 2000-hr durability (TBO).
- Altitude capability: 28 000 ft or more (depends on application).
- Modular design (family-of-engines concept).
- Manufacturing cost competitive with comparable reciprocating engines.

Specific objectives of the program, believed to be accomplishable by the end of 1991, are shown in the figure above.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Stratified-Charge Rotary Engine Concept



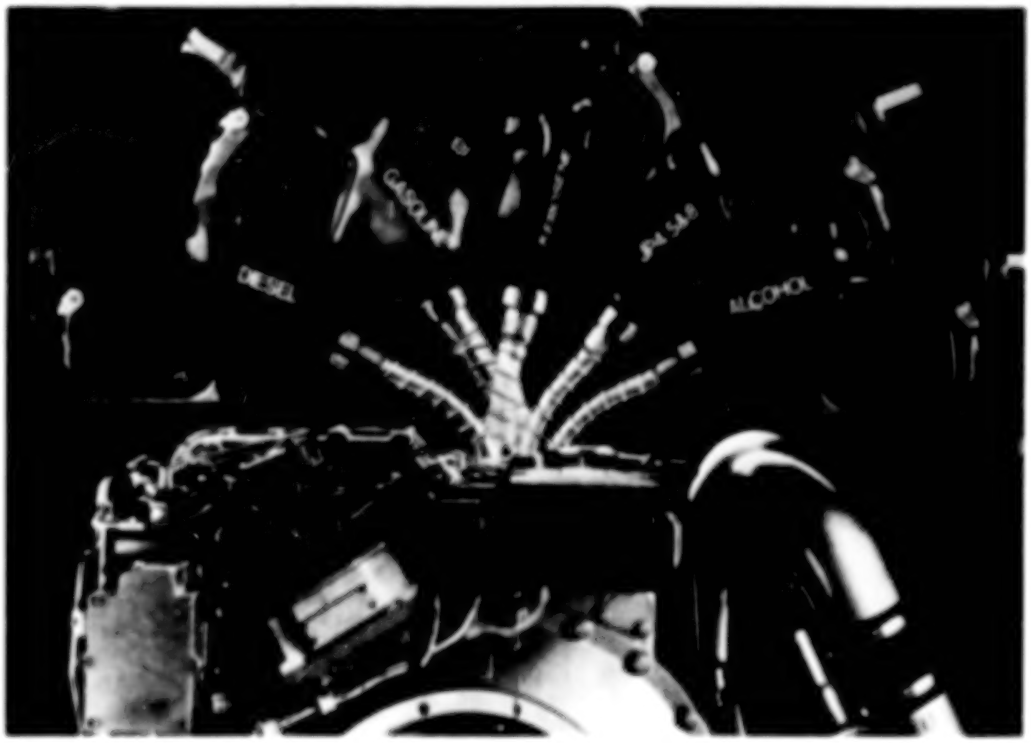
CD-81-63976

A particular form of the rotary engine, the so-called stratified-charge rotary engine (SCRE) was considered to have the best chance of meeting the objectives. In this embodiment (see chart), the primary structure consists of a triangular rotor which is given a hula-hoop type motion within a trochoidal housing. The rotor's motion, guided by an eccentric crank and timing gears, is such that the rotor makes one complete revolution for three revolutions of the crank, while its sealed tips maintain contact with the trochoid walls. Thus, each of the rotor flanks defines a sealed working space in which the familiar strokes of intake, compression, combustion/expansion, and exhaust are accomplished. One complete four-stroke cycle is accomplished per revolution of the crankshaft.

In common with other versions of the Wankel engine, the SCRE has clear virtues of compactness, low weight and vibration, high power and simplicity. The stratified-charge version, however, employs diesel-type, timed, high-pressure fuel injection, *across an ignitition source*, to maintain positive control of the combustion event. By appropriate control of the fuel injection and ignition timings, the combustion can be controlled to a point where cetane and octane requirements on the fuel are irrelevant. This immediately leads to multifuel capability. Clearly, it also leads to major challenges and opportunities in understanding and optimizing this form of combustion.

Prior Accomplishments (FY '89 and earlier)

Any fuel, anytime, in any combination



CD-91-53077

A program addressing the previously stated objectives for the SCRE has been underway since the mid-1980's and is now entering its final phase. The effort has been structured to consist of major R&D contracts with Curtiss-Wright and Deere, supported by smaller NASA in-house efforts and grants with several Universities.

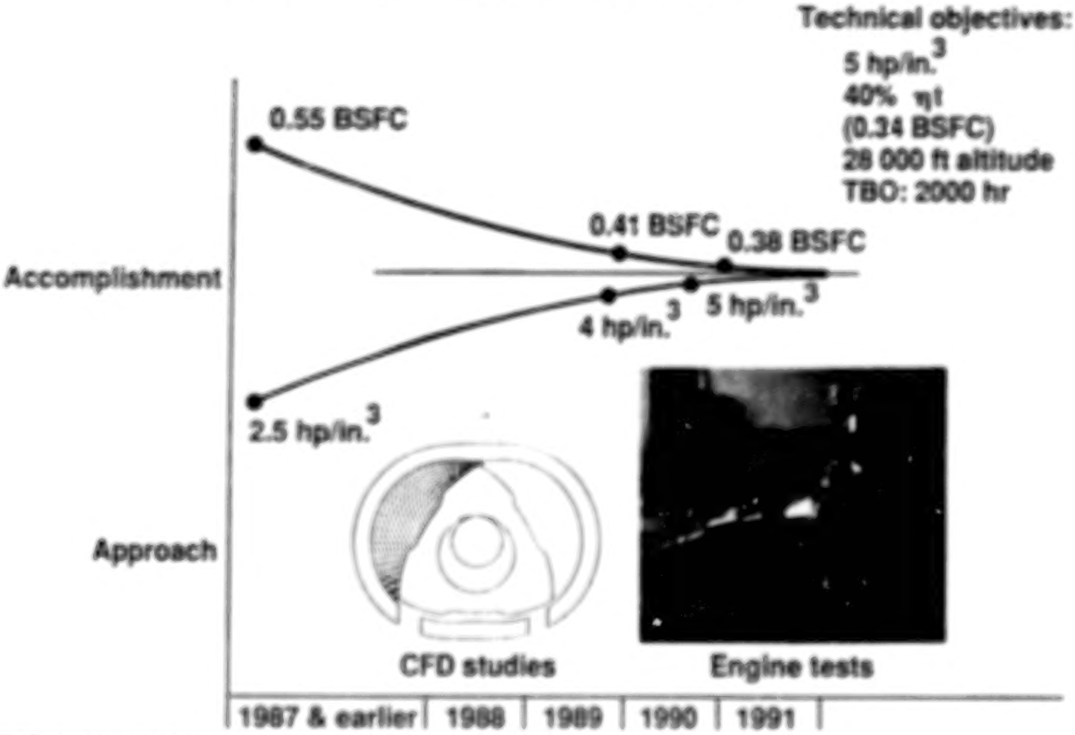
One of the major objectives, multifuel capability, has been demonstrated. Engine tests with automotive and aviation gasoline (avgas), diesel fuels, kerosene/Jet-A, JP4, 5, and 8, and alcohol fuels have shown virtually identical thermal efficiencies and no octane/cetane dependence. This is inherent in the stratified-charge combustion concept which is particularly well suited to the rotary engine. This alone makes it usable and thus, saleable, in parts of the world where specific fuels, e.g., avgas, may be unavailable or prohibitively expensive.

Major progress against other objectives was recorded as well. Specific power was nearly doubled, to above 4 hp/in.^{3} by the end of FY '89. At the same time, cruise BSFC was improved about 20 percent from its original baseline and had reached 0.42 lb/hp-hr by late 1989. These gains also demonstrated the power of CFD methods for improving combustion in the rotary engine and the ability of turbocharging to improve BSFC as well as power. One CFD-driven improvement, the so-called non-shadowing or fan type main fuel injector spray pattern, alone accounted for 14 percent out of the 20 percent BSFC improvement. The other 6 percent was due to turbo optimization.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Recent Accomplishments (Early '91)

Rotary Engine Performance Improvement



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CD-91-[illegible]

Following the precedent described above, the recent efforts have continued to rely heavily on further CFD-derived combustion system modifications and continued turbocharger matching and optimization. The so-called non-shadowing spray was further improved to achieve 0.41 BSFC by early 1990. A second CFD-related modification, the dual-spray pilot injector, experimentally yielded a 0.39 BSFC by January 1991. Most recently, the dual ignition housing design, which also derives from CFD studies, is currently under test and yielding BSFCs of around 0.38.

In addition, tests with higher levels of turbocharging yielded a maximum specific power output of 5.0 hp/in.^{3}, thus satisfying another of the original objectives. In the process, it was observed that the test engine was running richer than would be desired for good fuel economy. This indicates a need for more air throughput, which could be obtained either by increasing the exhaust port area or by further raising the turbocharger discharge pressure and flow. The former approach was selected since the tests also showed a marginal peak pressure condition at high powers.

Additional tests involved a ceramic-coated rotor developed in a companion SBIR effort. Short-term durability was demonstrated in 100 hr of running, and the heat transfer results appeared to be consistent with computer code predictions.

Other accomplishments include two new CFD codes and a new cycle/system simulation code now operational and in the validation process. Fuel spray and airflow visualizations are being conducted to obtain validation data. Two new apex seal material candidates (Si_3N_4 and a Deere proprietary sintered metallic) have demonstrated 6000-hr TBO potential.

Rotary Engine CFD Code Development and Applications

As previously noted, three-dimensional, turbulent, Navier-Stokes computer codes to simulate the airflow inside of a rotary engine have been under development for several years. These include appropriate submodels for fuel spray phenomena and combustion. Historically, the earliest and most extensively applied code is that developed at Princeton University and improved and extensively used at Deere. A comparable code, offering potential advantages in CPU speed, sensitivity, and user-convenience features, has been developed at Lewis by a NASA-Sverdrup team. This code has recently become operational and is now beginning the assessment/validation process.

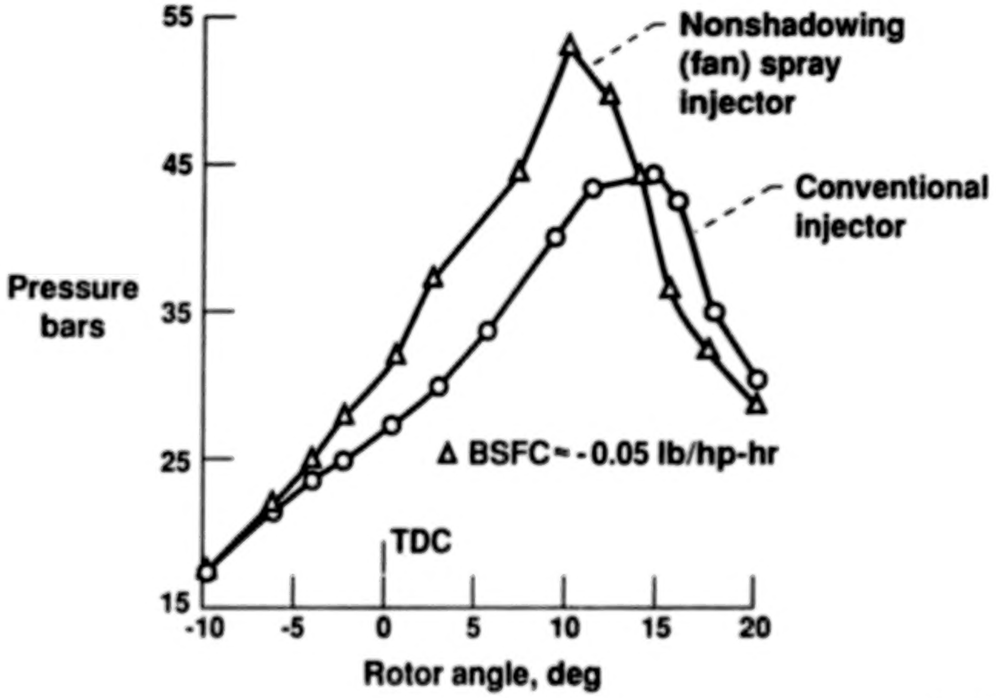
The availability of such codes now allows analytical studies to be made, in as much depth and detail as desired, of the airflow, fuel spray, evaporation, mixing, and combustion processes in the rotary engine. The accompanying chart illustrates typical gridwork, fuel droplet trajectories, temperature profiles, and fuel concentration near top-dead-center at a moderate speed/load condition.

Early studies consistently showed evidence of slow and nonuniform fuel-air mixing in the pocket region. The shadowing effect of one fuel jet directly upstream of another was one of the primary contributors to this problem. The solution - which seems almost obvious in retrospect - was to arrange the fuel sprays into a two-dimensional, fan-shaped pattern, rather than the diesel type conical spray that was used previously. In this arrangement (previously referred to as the non-shadowing spray), no fuel jet is the wake of an upstream one, and all receive the full momentum flux of the local airflow field.

ORIGINAL PAGE IS
OF POOR QUALITY

C 6

Rotary Combustion Improvements Resulting From CFD Work

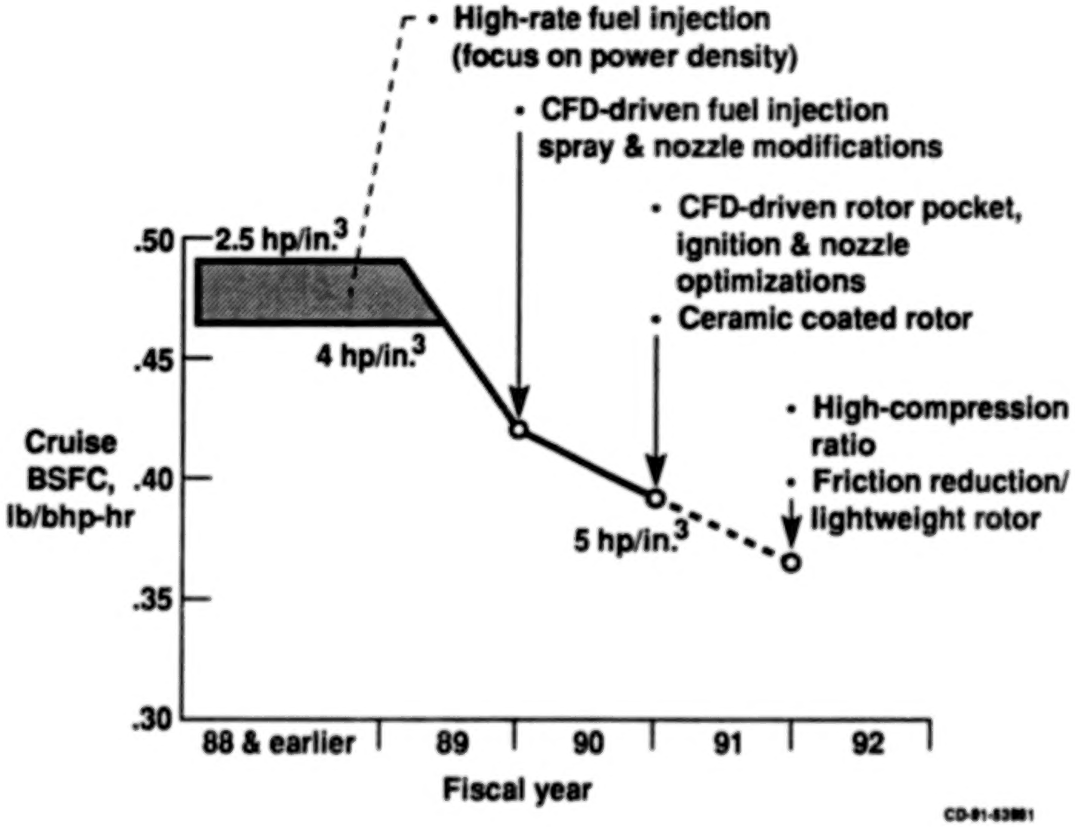


CD-91-53980

The advent of believable CFD methods has had a revolutionary impact on rotary engine combustion since about 1988. The difference - from the viewpoint of the rotary engine research teams - has been as if we were given a powerful searchlight instead of a single candle with which to explore a very large cave. The flow and combustion phenomena within the stratified-charge rotary are so unlike their diesel or turbine counterparts that prior experience with other heat engines provided no useful guidance whatsoever in the combustion area. On the contrary, the initial choice of a conventional, diesel type conical spray pattern for the main fuel injector proved to be exactly opposite to what the rotary engine really needs.

As previously mentioned, numerous improvements to the rotary combustion system have now been identified by the CFD studies. The first of these, the so-called non-shadowing, or fan spray, pattern for the main fuel injector nozzle, alone resulted in an *immediate* improvement of about 10 percent in cruise BSFC when tested in the engine rig. As illustrated in the accompanying chart, the fan spray resulted in faster fuel-air mixing and, in turn, a faster rate of pressure rise in the combustion chamber, a higher peak pressure, and a closer approach to the ideal Otto cycle. In the first experiments with this innovation the rig engine improved from a BSFC of 0.48 to 0.43 at cruise conditions within a few weeks. This is believed to be *the first time in history that a CFD-derived modification has led to an actual improvement in a real engine.*

Rotary Combustion Improvements (cont.)



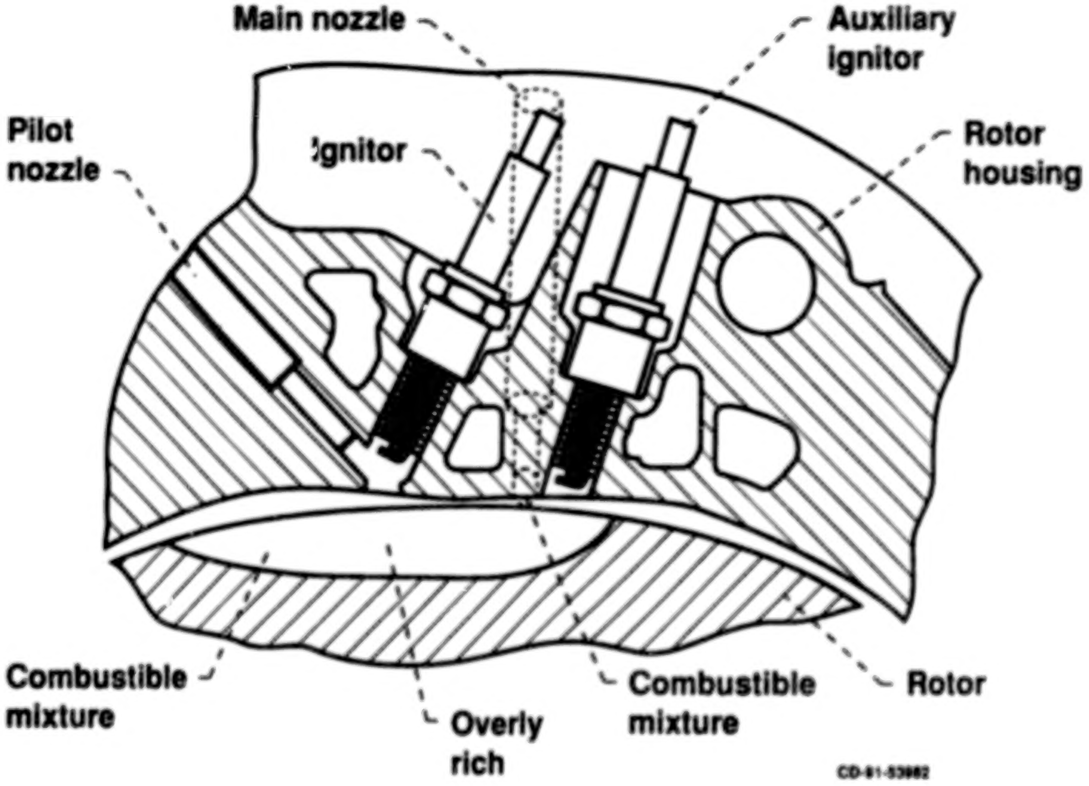
CD-91-53881

The chart summarizes the past and expected BSFC improvements. Following the major accomplishment described on the preceding page, the BSFC of the fan spray engine configuration was further improved to about 0.41 during 1990. This entailed experimental optimization of the engine/turbocharger match and injector details such as hole size, number, fan angle, and fuel pressure.

Additional tests during 1990 addressed the issue of maximum power density. The objective level of 5 hp/in.^{3} was achieved, albeit with some difficulty. This is a factor-of-two improvement over the original baseline and over contemporary automotive rotaries. During these tests, it was noted that the engine was unable to pass enough airflow to maintain near-optimum mixture ratios at high speeds and loads. A new housing with an enlarged and variable exhaust port area was designed to handle the increased airflow without higher boost pressures.

Other CFD-related improvements include the dual spray pilot injector, which demonstrated 0.39 BSFC in January 1991; the dual igniter housing, which is currently running at about 0.38 BSFC; the variable exhaust port housing, which should improve efficiency at the higher power levels; and two modified-pocket rotors (compact pocket and dual pockets), one or both of which should achieve the predicted 0.36 BSFC level. Finally, the time-honored thermodynamic expedient of increasing the compression ratio above its present, conservative value of 8.4:1 may well result in a cruise BSFC below 0.36.

Benefit of Dual Spark Ignition



CD-91-53982

The dual-spark igniter concept, illustrated schematically in the accompanying chart and currently under test at Deere, is one of the several improvements that have resulted from the CFD analyses. As the illustration suggests, the additional spark plug provides for immediate ignition of a secondary region of combustible mixture. Otherwise, flame propagation is retarded by an intervening layer of overly rich fuel-air mixture. This results in a faster pressure rise, higher peak pressure, and improved thermal efficiency, exactly as was previously shown for the fan spray injector. Currently, a BSFC of about 0.38 lb/hp-hr is being shown. Moreover, the benefit from this concept is over and above the benefits due to the fan injector and dual-spray pilot, although individual benefits are not expected to add together linearly (since they are collectively subject to an upper limit set by basic thermodynamics).

Perhaps the most promising of the planned modifications are those involving changes to the rotor's combustion pocket-size, shape, location, or any combination thereof. These were deferred until late in the program in order to develop confidence in the CFD results. New rotor castings are relatively expensive in small lots and involve substantial lead times. Nevertheless, it is expected that at least two new pocket configurations will be tested during FY '91 and will result in meeting the present cruise BSFC objective.

Issues and Problems

- **Deere & Co. plans to divest itself of the rotary engine division by November 1991.**
- **NASA response: Develop plans & contract modification to complete the Deere contract by November 1991.**

CD 91-53983

In September 1990, Deere and Company announced plans to divest itself by November 1, 1991, of its Rotary Engine Division (REDIV) in New Jersey, where the majority of the NASA rotary technology enablement program is being carried out. This announcement was accompanied by a substantial reduction in force and cessation of all company-sponsored efforts at the REDIV facility. Continuing efforts will address the completion of the NASA and other major contracts "to the customer's satisfaction."

This posed difficulties for NASA since the original program was planned and scheduled for completion at the end of FY '92. The unexpected acceleration of the program affects several Universities and small companies as well as Deere and NASA. NASA has responded by developing plans to complete a worthwhile subset of the original plans by November 1991. Nevertheless, an acceleration of one full year out of two years remaining cannot be accommodated without something being lost. The major loss as now perceived is that maximum power and best BSFC will not be obtainable in a single build of the test rig. At the end, there will be no engine which is capable of 5 hp/in.^{3} *and* a 0.35 to 0.36 cruise BSFC. A final step, which may well require a structural redesign for higher peak pressures, will still be needed. Also needed are tests of further known improvements, such as ceramic applications and turbocompounding, which could not be fitted into the compressed schedule.

Currently, the search for a successor is underway in earnest. Although the present authors are not privy to this process, we can certainly say that much will depend on the identity, objectives, and capabilities of the successor(s). At this writing we cannot predict whether the complete engine will ever be built, or by whom, or whether the additional known improvements will ever be tested.

Summary

BSFC Improvement steps - accomplished or planned

Original goals

Fan spray main injector	0.42/lb/hp-hr	1989 baseline
Dual-spray pilot	0.39 lb/hp-hr	Jan. 1991
Dual-spark housing	0.38 lb/hp-hr	Feb. 1991
Revised porting	0.37 lb/hp-hr	Mar. 1991
Higher compression	0.36 lb/hp-hr	May 1991
CFD-optimized pockets	0.35 lb/hp-hr	Sep. 1991

Future potential

Air-assist injection	0.34 lb/hp-hr
Friction reduction, Ti rotor, Si_3N_4 apex seal	0.33 lb/hp-hr
Turbocompounding	0.30–0.32 lb/hp-hr

CD-91-53984

Despite the issues just cited, it is clear that major and perhaps revolutionary progress has been accomplished in the NASA Rotary Engine program over the past few years. This includes multifuel capability, 5 hp/in.^{3}, a cruise BSFC rapidly approaching this year's goal of 0.35 to 0.36 lb/hp-hr and highly encouraging wear-rate indications for ceramic apex seals. Highly sophisticated new computer codes have been developed and applied to generate many of the specific improvements enumerated in the chart above.

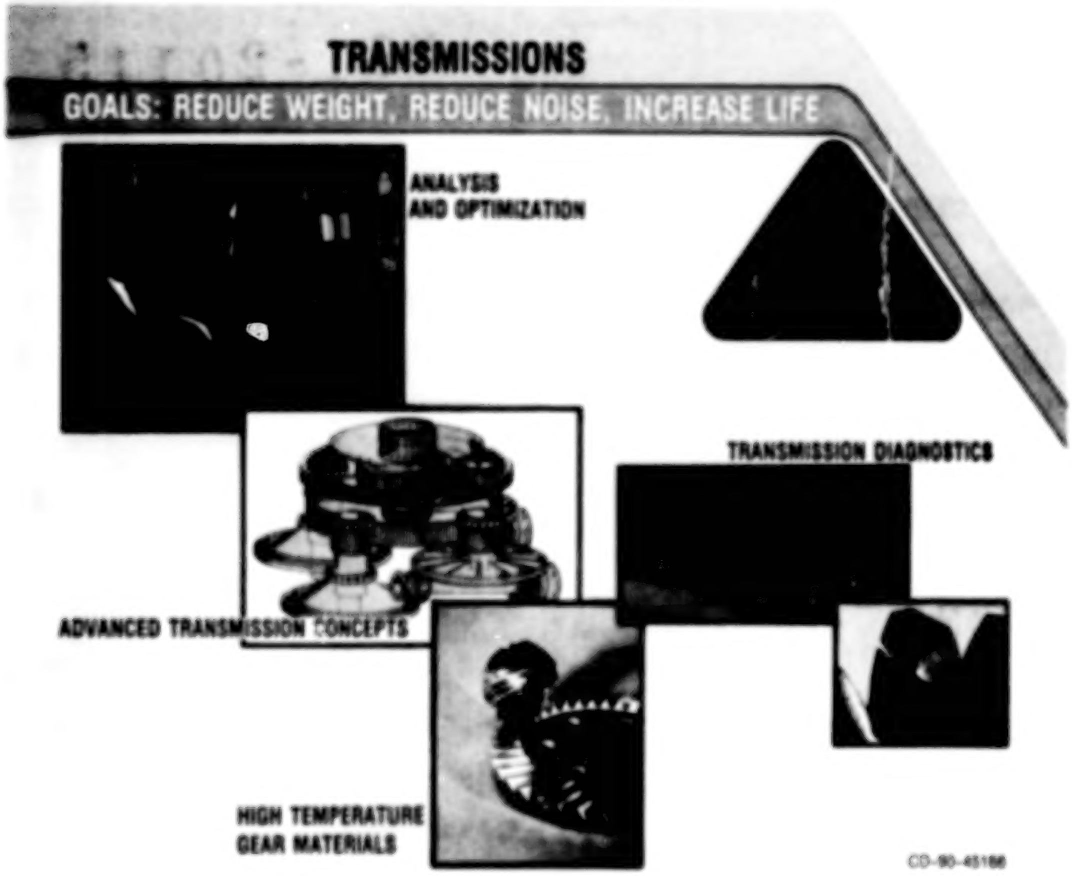
Because of the limited time, unfortunately, it now appears that the maximum-power and best-BSFC objectives will not be demonstrable in the same engine and that several known improvements cannot be exploited. Depending on who becomes the successor to Deere and on resources yet to be identified, the potential nevertheless exists to achieve a cruise BSFC as low as 0.30 lb in a compact, lightweight, multifuel rotary engine.

N91-20115

ADVANCED ROTORCRAFT TRANSMISSION TECHNOLOGY

David G. Lewicki
Propulsion Directorate
U.S. Army Aviation Systems Command
NASA Lewis Research Center
Cleveland, Ohio

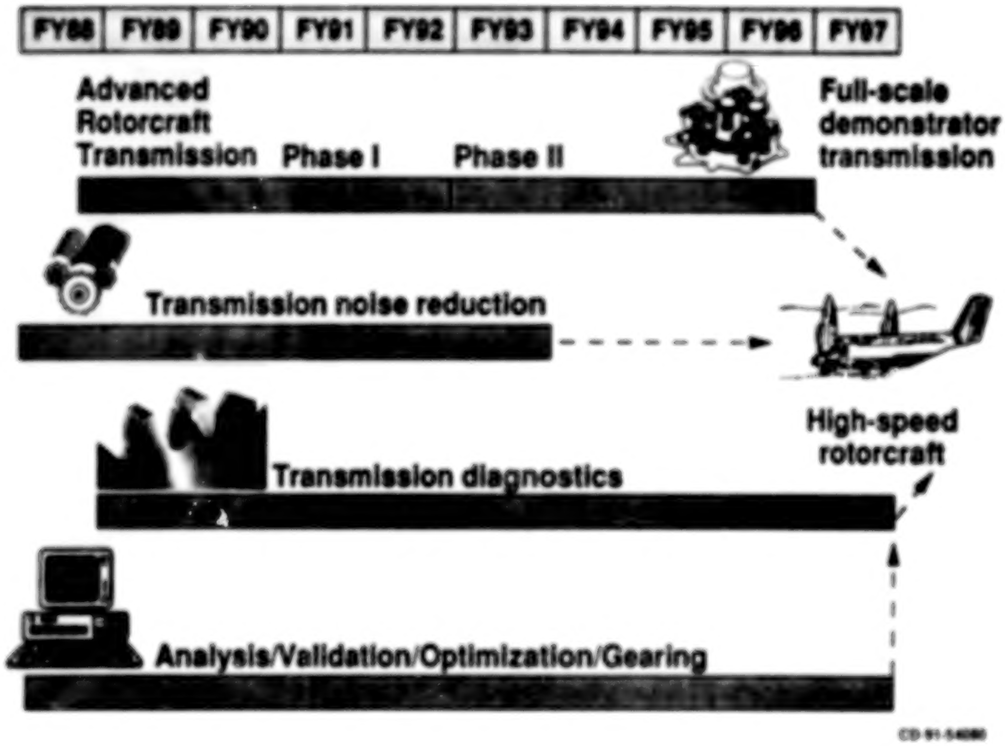
The NASA Lewis Research Center and the Propulsion Directorate of the U.S. Army Aviation Systems Command are involved in a joint research program to advance the technology of rotorcraft transmissions. The program consists of analytical and experimental efforts to achieve the overall goals of reducing transmission weight and noise, while increasing life and reliability. The work includes in-house studies and tests, university grants, industry contracts, and joint programs with other military organizations. This paper highlights recent activities in the areas of transmission and related component research. Three major activities are highlighted: the Advanced Rotorcraft Transmission (ART) program, a comprehensive transmission noise reduction program, and a transmission diagnostics research effort. Results of recent activities are presented along with future research plans.


TRANSMISSIONS
GOALS: REDUCE WEIGHT, REDUCE NOISE, INCREASE LIFE
ANALYSIS AND OPTIMIZATION
TRANSMISSION DIAGNOSTICS
ADVANCED TRANSMISSION CONCEPTS
HIGH TEMPERATURE GEAR MATERIALS
CD-90-45166

Joint rotorcraft transmission research began in 1970 with the NASA Lewis Research Center and the Propulsion Directorate of the U.S. Army Aviation Systems Command, and continues to the present (refs. 1 to 8). The major goals of the program are to increase the life, reliability, and maintainability; reduce the weight, noise, and vibration; and maintain the relatively high mechanical efficiency of the drive train in helicopter and turboprop transmissions. Lighter transmissions increase vehicle range, payload, and performance. Higher life and reliability produce safer operation and lower operating costs. Quieter transmissions increase pilot and passenger safety and comfort. The approach in achieving these goals is to identify advanced materials and lubrication schemes, as well as advanced design concepts and analyses for both transmission components and total transmission systems. The research efforts consist of in-house projects, university grant work, and industry-contracted efforts. Unique experimental testing facilities for transmissions exist at Lewis (ref. 9) for verifying analytical codes as well as demonstrating advanced design concepts and testing mechanical components, materials, and lubrication techniques.

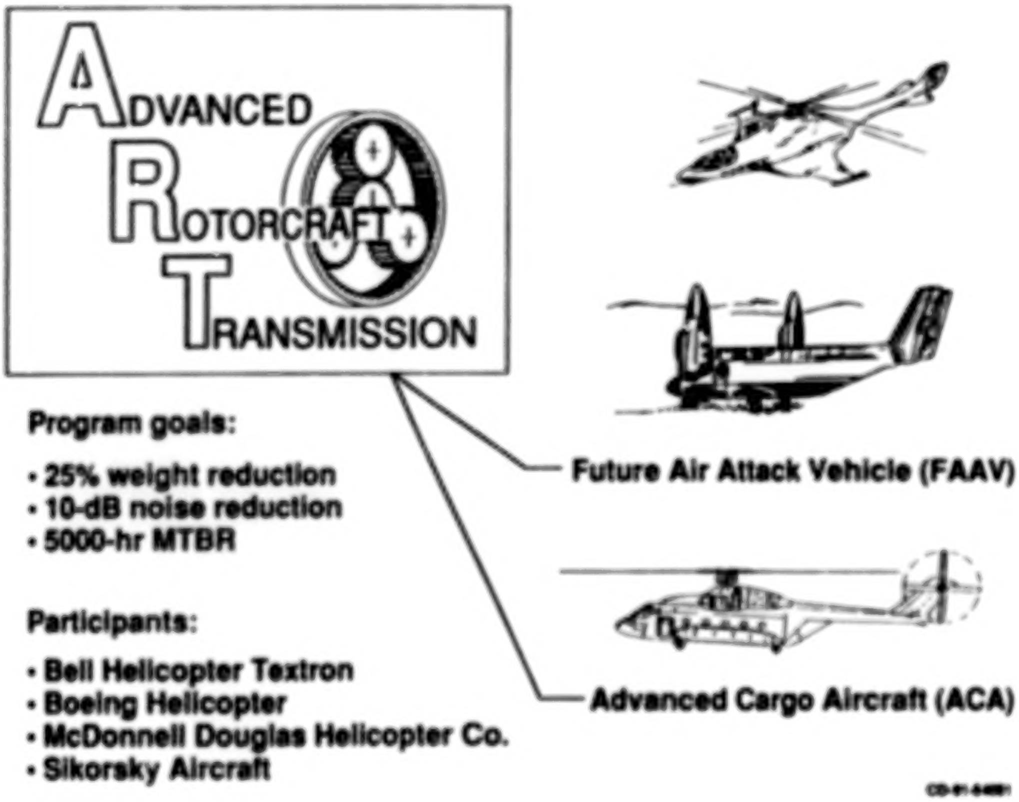
The purpose of this paper is to review the current helicopter transmission research at Lewis. Results of recent activities are presented along with future research plans. The emphasis of the reported work is on three current activities: (1) the Advanced Rotorcraft Transmission (ART) program, (2) a comprehensive transmission noise reduction program, and (3) a newly developed effort in transmission diagnostic research.

Transmission Research


FY88
FY89
FY90
FY91
FY92
FY93
FY94
FY95
FY96
FY97
Advanced Rotorcraft Transmission
Phase I
Phase II
Full-scale demonstrator transmission
Transmission noise reduction
Transmission diagnostics
High-speed rotorcraft
Analysis/Validation/Optimization/Gearing
CD-91-54280

The ART program is intended to develop and demonstrate lightweight, quiet, and reliable drive train systems for next-generation Army rotorcraft. This program is primarily a contracted effort with some Lewis in-house technical support and testing. The ART program started in fiscal year 1988 with phase I (Preliminary Design and Component Validation), and phase II (Full-Scale Demonstrator) should be completed by fiscal year 1997. The goal of the transmission noise reduction program is to reduce the gearbox noise by 10 dB in the next generation of transmissions for advanced rotorcraft. This effort, which started in 1988, is a mixture of university grant activities and in-house analysis and experimental studies. It is anticipated that by 1994 an acoustic computer code for transmissions will be developed and validated. Transmission diagnostic research at Lewis is a rather new effort which started around 1989. The objective is to evaluate and develop diagnostic methods for early detection of incipient failures of critical drive train components. The long term goal is to demonstrate a transmission health and monitoring system by 1998.

The research efforts described above are complemented with basic research projects in gearing, analysis and validation of transmission components, and optimization. A unique data base has been established using the NASA Spur Gear Fatigue Test Apparatus, supplying valuable data on gear materials, lubrication, and life analysis. Cooperative programs with other military organizations exist, such as the Navy/NASA/Army Advanced Lubricants Program, with the goal of developing a separate transmission lubricant with improved load-carrying capacity, higher temperature capabilities, and corrosion resistance. In addition, analytical codes are being developed for gear, bearing, and transmission systems in the areas of dynamics, life, and mechanical performance.

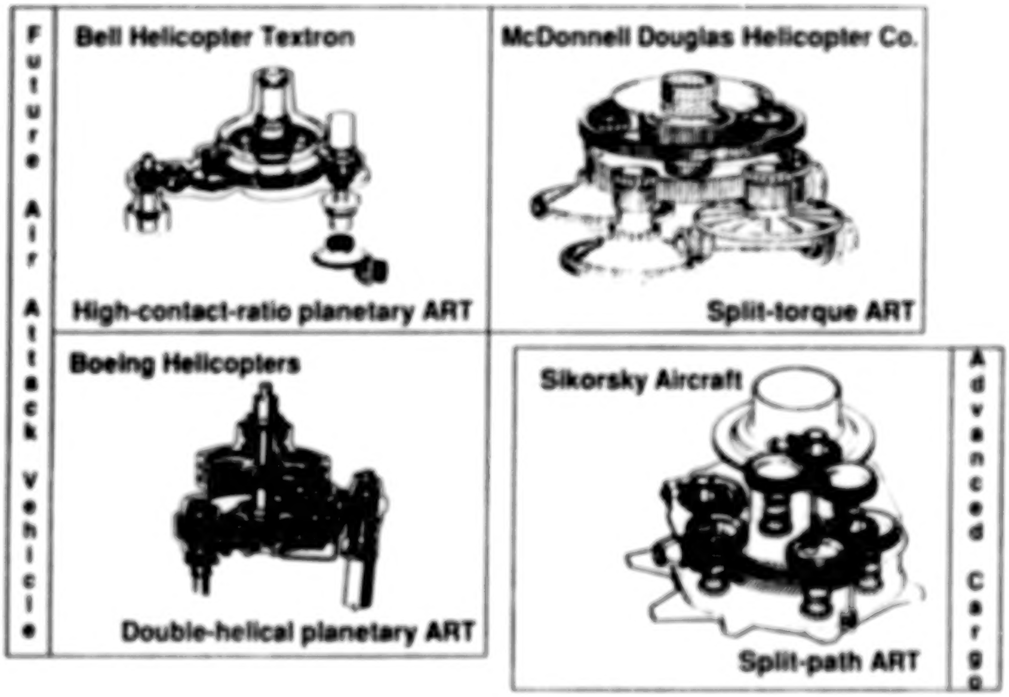

ADVANCED ROTORCRAFT TRANSMISSION
Program goals:
• 25% weight reduction
• 10-dB noise reduction
• 5000-hr MTBR
Participants:
• Bell Helicopter Textron
• Boeing Helicopter
• McDonnell Douglas Helicopter Co.
• Sikorsky Aircraft
Future Air Attack Vehicle (FAAV)
Advanced Cargo Aircraft (ACA)

The Advanced Rotorcraft Transmission (ART) program is an Army funded, joint Army/NASA program. Its intent is to develop and demonstrate lightweight, quiet, durable drive train systems for next generation rotorcraft (ref. 10). ART addresses general requirements of two distinct next-generation aircraft classes: (1) the Future Air Attack Vehicle, a 10 000- to 20 000-lb aircraft capable of undertaking tactical support and air-to-air missions, and (2) the Advanced Cargo Aircraft, a 60 000- to 80 000-lb aircraft capable of heavy-lift field support operations. The ART program includes both tiltrotor and more conventional helicopter configurations. Its specific objectives include reducing the drive train weight by 25 percent (compared with baseline state-of-the-art drive systems configured and sized for next-generation aircraft), reducing the noise level at the transmission source by 10 dB relative to the suitably sized and configured baseline, and increasing mean time between removal (MTBR) to 5000-hr.

ART consists of two phases. The first is the Preliminary Design and Component Validation phase. Here, new ideas in gear configuration, transmission concepts, and airframe/drive train integration are being studied. Also included is the application of the latest available component, material, and lubrication technologies. Contract participants include McDonnell Douglas Helicopter Company, Boeing Helicopters, Bell Helicopter Textron, and Sikorsky Aircraft. The Lewis Research Center is also providing in-house technical support and testing. The second phase of the ART program is the Full-Scale Demonstrator phase where a full-size version of the transmission designated in phase I will be designed in detail and tested to demonstrate the goals of the program.

Advanced Rotorcraft Transmission Program

Advanced Transmission Configurations


Future Air Attack Vehicle
Bell Helicopter Textron
High-contact-ratio planetary ART
McDonnell Douglas Helicopter Co.
Split-torque ART
Boeing Helicopters
Double-helical planetary ART
Sikorsky Aircraft
Split-path ART
Advanced Cargo

All four ART contractors have selected their vehicle application, baseline aircraft, and baseline transmission. Candidate drive train systems have been developed to a conceptual design stage. Tradeoff studies based on comparative weight, noise, and reliability have resulted in selection of the ART transmission for each of the four participating contractors. Preliminary designs of each of the four selected ART transmissions have been completed, as have mission impact studies. Comparisons of aircraft mission performance and life cycle costs were undertaken for next-generation aircraft with ART and with the baseline transmission.

McDonnell Douglas Helicopter Company (MDHC) selected the Future Air Attack Vehicle (FAAV). They defined their baseline as an upgraded AH-64 Apache helicopter and chose a split-torque transmission with face gears as their advanced technology configuration (ref. 11). Boeing Helicopters chose the FAAV application and a tilt-rotor aircraft as their baseline. They selected a helical three-stage planetary configuration for ART (ref. 12). Bell Helicopters also selected the FAAV application and a tilt-rotor aircraft as their baseline, but they chose a high-contact ratio planetary configuration for ART (ref. 13). Sikorsky Aircraft was the only participant to select the Advanced Cargo Aircraft application. They chose an upgraded CH-53 helicopter as their baseline and a split-path configuration for ART (ref. 14). All four contractors have projected that the ART goals for weight, noise, and life are attainable. All contractors deduced from the mission analysis that huge savings in life cycle costs could be achieved if an advanced transmission design is incorporated in the aircraft during development of the airframe.

Advanced Rotorcraft Transmission Program

Planned Component Testing

Milestones completed:

- Key component technologies identified
- Component test plans developed

Contractor Tests

- 1/2 size split-path transmission
- Planetary tests
- Gear tests
- Bearing tests
- Materials tests

ADVANCED
ROTORCRAFT
TRANSMISSION

Lewis in-house tests

- Low-noise spur and spiral bevel gear tests
- Face gear tests

Significance:

- Identification permits testing to demonstrate capability to attain weight, noise, and life goals.

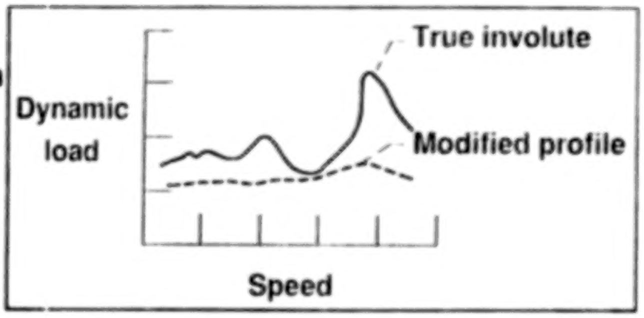
CD-91-[illegible]

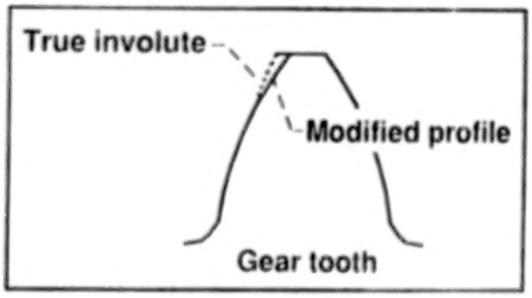
For these ART configurations to meet the program goals, certain critical component technologies must be incorporated into the preliminary designs. MDHC plans to perform single-tooth bending tests, gear scoring tests, impact toughness tests, and fracture toughness tests for a variety of advanced gear materials manufactured using near-net-shape gear forging to improve strength and reduce manufacturing costs. In addition, Lewis and MDHC have established a joint program to test face gears in the NASA Spiral Bevel Rig. Boeing Helicopters plans to test gears made from Vasco X-2 high-temperature material and manufactured using near-net-shape forging. They also plan to test lightweight titanium accessory drive gears with treated surfaces. Lewis and Boeing will jointly evaluate noise generation properties of eight parallel-axis gear configurations in the NASA Gear Noise Test Rig. In addition, Boeing has planned substantial hybrid bearing and active noise cancellation tests. Bell Helicopter is concentrating their component test program in three major areas: planetary gear train tests, material tests, and spiral bevel gear tests. The planetary gear train will be tested for noise and vibration, efficiency, fatigue, and loss of lubricant. Lewis and Bell have established a joint program to test spiral bevel gears in the NASA 500-hp Test Stand. Sikorsky Aircraft will test a scaled-down version (one-half size) of the split-path transmission configuration proposed for the Advanced Cargo Aircraft. Included in the testing are the following: high-contact-ratio double-helical gears on the output stage, double-helical pinions with large gear tooth face width-to-diameter ratios, an elastomeric torque-sharing device, topological gear tooth profiling for improved load distribution, high-temperature operation (300 °F), scoring-resistant X-53 gear material, and transmission error measurements.

Gear Tooth Profile Studies

Milestones completed

- **Comprehensive analytical study of effect of gear tooth profile on dynamic loads completed.**
- **Experimental verification plan established; test gears ordered.**





Significance:

- **Slight modification of gear tooth profile produces significant change in dynamic load.**
- **Dynamic loads can be significantly reduced through optimized gear tooth profiles.**

CD-91-54084

Helicopter noise and vibration are important topics because of health and environmental concerns, passenger comfort, and pilot efficiency. As power increases so does the weight penalty associated with cabin soundproofing materials. These facts have led to an effort to reduce noise and vibration at their origin. Today the main source of noise in helicopter interiors is the gear mesh in the transmission. The goal of the NASA Lewis transmission noise reduction program is to reduce the gearbox noise by 10 dB in next-generation transmissions for advanced rotorcraft (ref. 15).

Gear dynamic loading in rotorcraft transmissions is a major source of noise. Gear dynamic loading is caused by a complex interaction between the various gears in mesh and also the dynamic properties of the bearings, shafts, housings, and masses in the system. Gear dynamic load is caused by the unsteady relative angular motion of gear pairs. Not only is it a major source of noise, but it can significantly reduce component life. The computer program DANST (Dynamic ANalysis of Spur gear Transmissions) was used to investigate the effects of both linear and parabolic tooth profile modifications on the dynamic response of low-contact-ratio spur gears (refs. 16 and 17) and high-contact-ratio gears (refs. 18 and 19). The dynamic loading response of unmodified (perfect involute) gear pairs was compared with that of gears with various profile modifications. The effects of the total amount of modification and the length of the modification were studied at various loads and speeds to find the optimal profile to minimize dynamic loading and noise. Then, normalized design charts were generated for gear systems operating at various loads and tooth modifications. The amount and type of tooth profile modification had a significant effect on the predicted dynamic response.

Gear Noise Rig–Facility Design and Installation

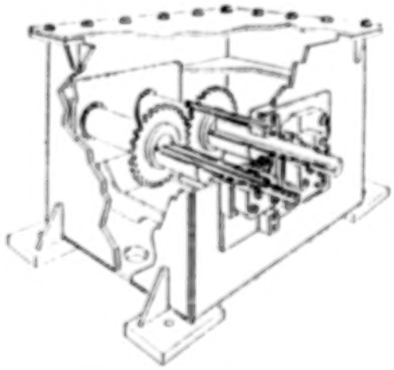
Gear Noise Rig

Milestones completed:

- Design and fabrication of highly instrumented test gearbox
- Test rig designed and constructed
- Facility is fully operational

Significance:

- Parametric studies of gear design variables
- Verification of computer models for gear dynamics and noise
- Technology base for future quiet transmission design



Test Gearbox

CD-91-54085

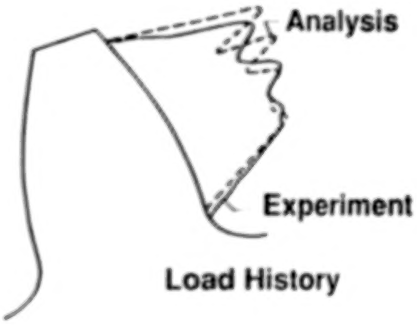
As previously mentioned, extensive studies have been performed to reduce gear dynamic loading, and consequently, lower overall transmission noise. Work has also resulted in the development of several computer codes and analyses. A finite element model of a geared rotor system with flexible bearings was developed (ref. 20). The program is capable of calculating the natural frequencies, corresponding mode shapes, and the dynamic loads at various positions in the system when excited by mass imbalance, geometric eccentricities, and transmission errors. An understanding of the interaction between the gear mesh and housing dynamics is also a crucial step in the overall plan of reducing vibration and noise in rotorcraft transmissions. One study analyzed the dynamics of the gear housing from the NASA Gear Noise Rig using both finite element analysis and experimental modal analysis (ref. 21). Another study concentrated on the complex transmission of vibration through rolling element bearings and resulted in the development of a new methodology for modeling transmissibility of vibration (ref. 22). In addition, the acoustic intensity program BEMAP (Boundary Element Method for Acoustic Prediction) was adapted to predict the sound field of a gearbox (ref. 23).

These new computer codes and analyses will form an important design tool for meeting the noise reduction goals. As a method to verify the codes, the NASA Gear Noise Rig was recently installed at Lewis. This rig will allow studies of gear design parameters and their effect on dynamic loads, vibration and noise production, propagation, and radiation. It will measure the effects on dynamic loads and noise caused by tooth profile, tooth spacing errors, gear misalignment, and damping.

Gear Dynamics Code Validation

Milestones completed:

- Compared analysis and experiment
- 28 test conditions
- Successfully predicts dynamic loads
- NASA TM-103232



Significance:

- Experimental verification of gear dynamics code
- Allows tuning gear profile for minimum noise and vibration

CD-91-54086

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

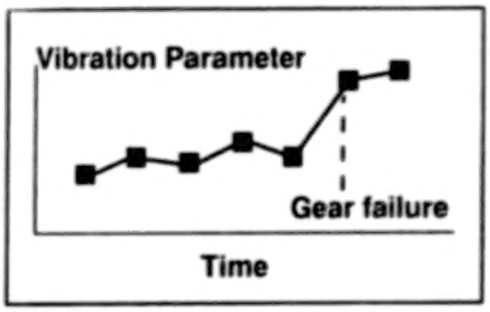
Gear vibration is primarily caused by the abrupt transition of the load from tooth to tooth as the gears mesh. The profiles of gear teeth are often modified to help smooth this dynamic action. The effects of tooth profile modification on gear dynamics is simulated by several computer codes developed through NASA grants and contracts. There has been an acute need to verify the gear dynamic load prediction codes with experimental data taken under carefully controlled conditions. The goal of this initial effort was to verify predictions of the gear dynamics code DANST for both the gear tooth loads and for the bending stress at the gear tooth root. Gear dynamics experiments were conducted in the NASA Gear Noise Rig. These tests were completed on a pair of standard involute spur gears with a linear profile modification run at a variety of speed and load conditions (ref. 24). Strain gages were used to measure the dynamic force acting between gear teeth, and the experiments were compared with the analysis.

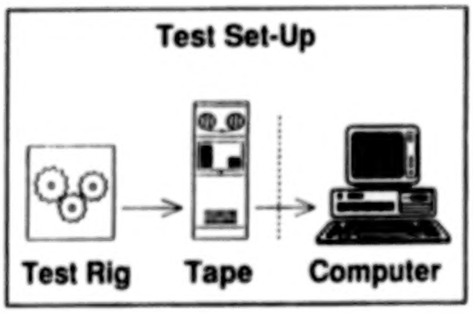
The maximum dynamic load prediction from DANST generally agreed with measurement results within 6 percent. The bending stresses predicted from DANST generally agreed with measurement results within 10 to 15 percent. The analysis also successfully predicted loss of tooth contact that occurred when the gears were run at high speeds and with a light load. The validated code will thus allow a gear designer to tune the profile of a new gear to minimize dynamic loads and stress, with the potential to reduce noise and vibration without expensive and time-consuming hardware testing. The code will also predict the effects of load and speed on the dynamics of a gear system.

Gear Failure Prediction Studies

Milestones completed:

- Vibration signatures of various modes of gear failures collected.
- Signatures analyzed using several failure prediction techniques.





Results:

- Algorithms to predict gear failures identified.
- Gear, bearing, and housing modal properties affected results.

CD-91-54087

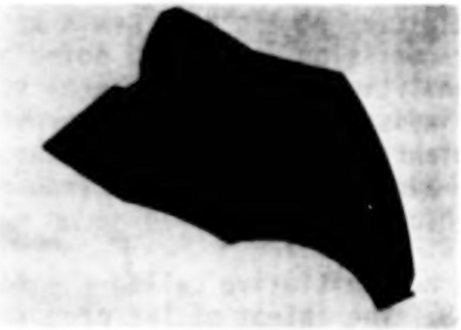
There has been a recent interest in transmission diagnostics for both safety and economy. A comprehensive transmission diagnostics program, of both basic and applied research, is being developed at Lewis. The basic research consists of evaluation and development of new and improved transmission component failure prediction techniques. The applied research consists of evaluating existing commercially available diagnostic devices by using the unique facilities at Lewis.

As an initial study, vibration signatures from 11 gear sets that were run from start to failure were collected from instrumentation installed on a single mesh gear fatigue test stand (ref. 25). Several gear mesh failure prediction methods were applied to the data to determine if a correlation existed between the various techniques and the observed modes of failure. The gear failures consisted of heavy wear and scoring, tooth breakage, single tooth pitting, and distributed pitting over several teeth. Results showed that the prediction methods were able to detect those gear failures which involved heavy wear or distributed pitting. Also, the modal frequency response between the gear shaft and transducer was found to significantly affect the vibration signal.

Future plans include installation of an on-line computer-based system on the single-mesh gear fatigue test stand and the 500-hp Transmission Test Stand. The objective will be to continue to evaluate the most promising gear failure detection algorithms and to develop new methods where needed. Oil debris monitoring methods will also be investigated as a diagnostic tool.

Spiral Bevel Gear Analysis and Experiments

Bevel Gear Rig


Computer Model

Test Specimen

- Tooth geometry and kinematic analysis
- Tooth contact analysis
- Finite element analysis
- Fatigue testing
- Noise/vibration testing
- Lubrication testing

CD-91-54088

In addition to the ART program, the transmission noise reduction program, and the diagnostics research, there are basic research efforts in gearing analysis and validation. One such effort is the work on spiral bevel gears. Advanced analytical capabilities such as finite element analysis are becoming commonplace in the design and development of components for next-generation rotorcraft. However, such techniques are not very straightforward for spiral bevel gears because of the complex tooth geometry considerations. Methods are being developed to use the gear geometry results for spiral bevel gears and merge the data with the current finite element code (ref. 26). The tooth coordinates and loading are input to a commercially available solid modeling package, PATRAN, for pre- and post-processing. Currently, the finite element analysis code NASTRAN is being used. The result is an in-depth stress and deflection analysis for spiral bevel gears, a valuable tool for the designer.

To verify analytical predictions and to improve the basic understanding of spiral bevel gear operation, the NASA Spiral Bevel Gear Rig is being used. Currently, the facility is being configured to conduct transient temperature tests. These tests will closely examine lubrication parameters such as oil temperature, flow, and oil jet orientation and will determine their effect on gear surface temperature. Critical parameters regarding efficient lubrication of high-speed spiral bevel gear systems will be investigated and identified. After these tests, material effects on spiral bevel gear fatigue life will be studied. Spiral bevel gears manufactured from six different materials are available for testing. The results of these tests will be compared with the extensive data base collected for spur gears to see if there is a correlation between parallel axis and spiral bevel gear life experiments.

CONCLUDING REMARKS

The purpose of this paper was to review the significant accomplishments of the past few years in helicopter transmission technology that have resulted from NASA/Army research activities at the NASA Lewis Research Center. Research program goals were to reduce transmission weight and noise and to increase life and reliability. A variety of analytical and experimental studies from in-house activities, university grants, and industry contracts were presented. Significant transmission weight reduction potentials were identified through the use of novel configuration arrangements and/or advanced components. Transmission life improvement potentials were identified through the use of advanced materials, lubrication, and component design.

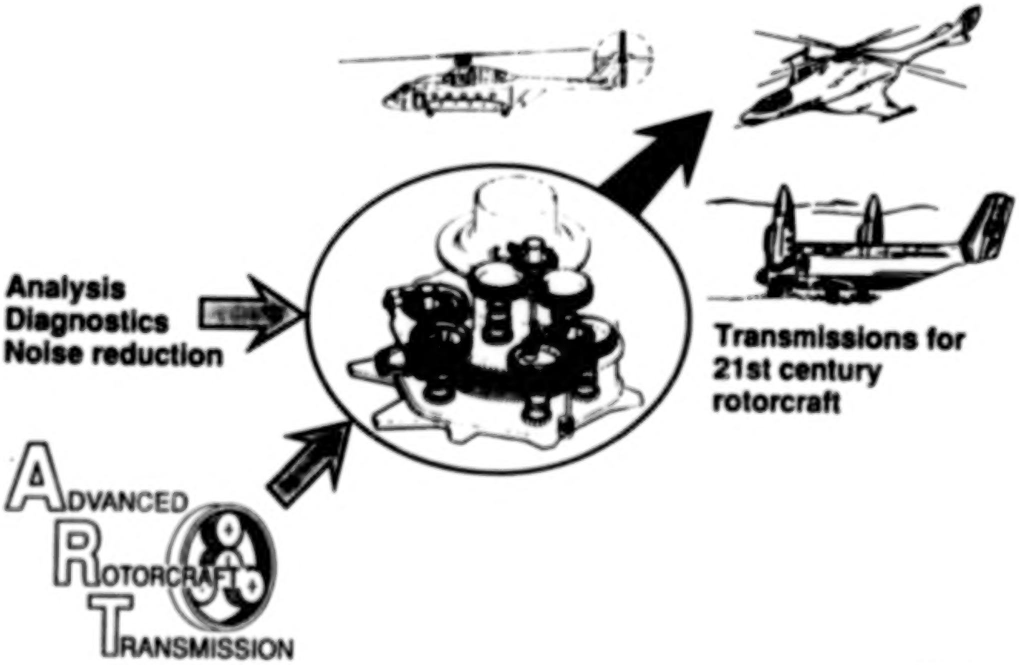
An important new initiative called the Advanced Rotorcraft Transmission (ART) program began in 1988. The intent of the program is to develop advanced concept transmissions for two categories of future Army rotorcraft: the Future Air Attack Vehicle and the Advanced Cargo Aircraft. The program parallels the concept offered by engine demonstration programs and will provide a way for the industry to develop advanced concepts and trial designs well in advance of critical needs.

An initial finding of the Advanced Rotorcraft Transmission (ART) program was that a generalized, comprehensive, noise prediction methodology for transmissions did not exist at the time of the configuration tradeoff studies. During these studies, baseline transmission noise data were used as a starting point. Extrapolation to each of the ART candidates was accomplished by comparing number of gear meshes, mesh loading, gear geometry features, and mesh speeds with those of the baseline. The result was typically a very qualitative comparison with a net judgment for each candidate as to whether it would be better, worse, much better, much worse, or the same as the baseline. Goals of the ongoing NASA transmission noise reduction program are to increase the basic understanding of the complex noise generation and transmissibility phenomena and to develop accurate, analytical tools for noise prediction. This will aid in the overall goal of reducing transmission noise and become a basis for next-generation design.

Transmission and drive train problems are major factors in vehicle maintenance costs and are a significant and costly cause of rotorcraft accidents, fleet groundings, and mission aborts. Drive trains must be periodically inspected, requiring a major disassembly of the transmission for close inspection. Transmission health monitoring can improve the safety of helicopter flight and improve maintenance efficiency. The Lewis diagnostics research combines analytical and experimental efforts to evaluate and develop methods for early detection of component failures.

It is reasonable to expect that rotorcraft will continue to evolve in the future. To achieve the necessary advances in rotary wing flight capabilities, drive train technology must keep pace with advances in engines, controls, structures, and rotors. The current plan for NASA/Army transmission research calls for a continued reduction in weight and noise, an increased emphasis in diagnostics, and continued support of military and civilian aviation activities. The overall intent is to provide technology readiness in transmissions for 21st century high-speed rotorcraft.

Summary



Analysis
Diagnostics
Noise reduction

Transmissions for
21st century
rotorcraft

ADVANCED
ROTORCRAFT
TRANSMISSION

CD-91-54089

REFERENCES

1. Zaretsky, E.V.; Townsend, D.P.; and Coy, J.J.: NASA Gear Research and Its Probable Effect on Rotorcraft Transmission Design. NASA TM-79292, 1979.

2. Zaretsky, E.V.; Coy, J.J.; and Townsend, D.P.: NASA Transmission Research and Its Probable Effects on Helicopter Transmission Design. NASA TM-83389, AVRADCOM-TR-83-C-3, 1983.

3. Advanced Power Transmission Technology. G.K. Fischer, ed., NASA CP-2210, AVRADCOM-TR-82-C-16, 1983.

4. Weden, G.J.; and Coy, J.J.: Summary of Drive-Train Component Technology in Helicopters. NASA TM-83726, USAAVSCOM-TR-84-C-10, 1984.

5. Coy, J.J.; Townsend, D.P.; and Zaretsky, E.V.: Gearing. NASA RP-1152, AVSCOM-TR-84-C-15, 1985 (Chapter 38 in Mechanical Design and Systems Handbook, H. Rothbart, ed., McGraw-Hill, 1985).

6. Coy, J.J.: Rotorcraft Transmission. Aeropropulsion '87, E.A. Willis, Jr., ed., NASA CP-3049, 1987, pp. 303-314.

7. Coy, J.J.; and Bill, R.C.: Advanced Transmission Studies. NASA TM-100867, AVSCOM-TR-88-C-002, 1988.

8. Lewicki, D.G.: Transmission Research Activities at NASA Lewis Research Center. NASA TM-103132, AVSCOM-TM-90-C-006, 1990.

9. Lewicki, D.G.; and Coy, J.J.: Helicopter Transmission Testing at NASA Lewis Research Center. NASA TM-89912, AVSCOM-TR-87-C-10, 1987.

10. Bill, R.C.: Advanced Rotorcraft Transmission Program. NASA TM-103276, 1990.

11. Bossler, R.B.; and Heath, G.F.: Advanced Rotorcraft Transmission (ART) Program Status. Presented at the American Helicopter Society Rotary Wing Propulsion Specialists Meeting, Williamsburg, VA, Nov. 13-15, 1990. Preprint No. 4.

12. Lenski, J.W.: Boeing Helicopters Advanced Rotorcraft Transmission (ART) Program Status. Presented at the American Helicopter Society Rotary Wing Propulsion Specialists Meeting, Williamsburg, VA, Nov. 13-15, 1990. Preprint No. 2.

13. Henry, Z.S.: Preliminary Design and Analysis of an Advanced Rotorcraft Transmission. Presented at the American Helicopter Society Rotary Wing Propulsion Specialists Meeting, Williamsburg, VA, Nov. 13-15, 1990. Preprint No. 3.

14. Kish, J.G.: Advanced Rotorcraft Transmission (ART) Program Review. Presented at the American Helicopter Society Rotary Wing Propulsion Specialists Meeting, Williamsburg, VA, Nov. 13-15, 1990. Preprint No. 5.

15. Zakrajsek, J.J., et al.: Gear Noise, Vibration, and Diagnostic Studies at NASA Lewis Research Center. NASA TM-102435, AVSCOM-TR-89-C-020, 1990.

16. Lin, H.H.; Townsend, D.P.; and Oswald, F.B.: Profile Modification to Minimize Spur Gear Dynamic Loading, NASA TM-89901, 1988.

17. Lin, H.H.; Oswald, F.B.; and Townsend, D.P.: Dynamic Loading of Spur Gears With Linear or Parabolic Tooth Profile Modifications. NASA TM-101444, AVSCOM-TR-88-C-003, 1989.

18. Lee, C., et al.: Influence of Linear Profile Modification and Loading Conditions on the Dynamic Tooth Load and Stress of High Contact Ratio Gears. NASA TM-103136, AVSCOM-TM-90-C-004, 1990.

19. Lin, H.H., et al.: Computer-Aided Design of High-Contact-Ratio Gears for Minimum Dynamic Load and Stress. NASA TM-103275, 1991.

20. Kahraman, A., et al.: Dynamic Analysis of Geared Rotors by Finite Elements. NASA TM-102349, AVSCOM-TM-89-C-006, 1990.

21. Lim, T.C.; Singh, R.; and Zakrajsek, J.J.: Modal Analysis of Gear Housing and Mounts. NASA TM-101445, AVSCOM-TR-88-C-041, 1989.

22. Singh, R.; and Lim, T.C.: Vibration Transmission Through Rolling Element Bearings in Geared Rotor Systems. NASA CR-4334, AVSCOM-TR-90-C-019, 1990.

23. Seybert, A.F.; Wu, T.W.; Wu, X.F.; and Oswald, F.B.: Acoustical Analysis of Gear Housing Vibration. NASA TM-103691, 1991.

24. Oswald, F.B., et al.: Comparison of Analysis and Experiment for Dynamics of Low-Contact-Ratio Spur Gears. NASA TM-103232, AVSCOM-TR-90-C-017, 1991.

25. Zakrajsek, J.J.: An Investigation of Gear Mesh Failure Prediction Techniques. NASA TM-102340, 1989.

26. Handschuh, R.H.; and Litvin, F.L.: A Method for Determining Spiral Bevel Gear Tooth Geometry For Finite Element Analysis. NASA TP-3096, 1991.

N91-20116

OVERVIEW OF SUBSONIC TRANSPORT PROPULSION TECHNOLOGY

John F. Groeneweg
NASA Lewis Research Center
Cleveland, Ohio

This overview summarizes the major elements of the NASA Lewis Research Center's subsonic transport propulsion research program. Ultra-high-bypass ratio cycles and high-efficiency cores are being investigated for propulsive and thermal efficiency improvements, respectively. Overall efficiency gains are sought subject to the constraints of noise and emissions goals. Elements of the research program including key technical issues are discussed along with the planned sequence of the activities.

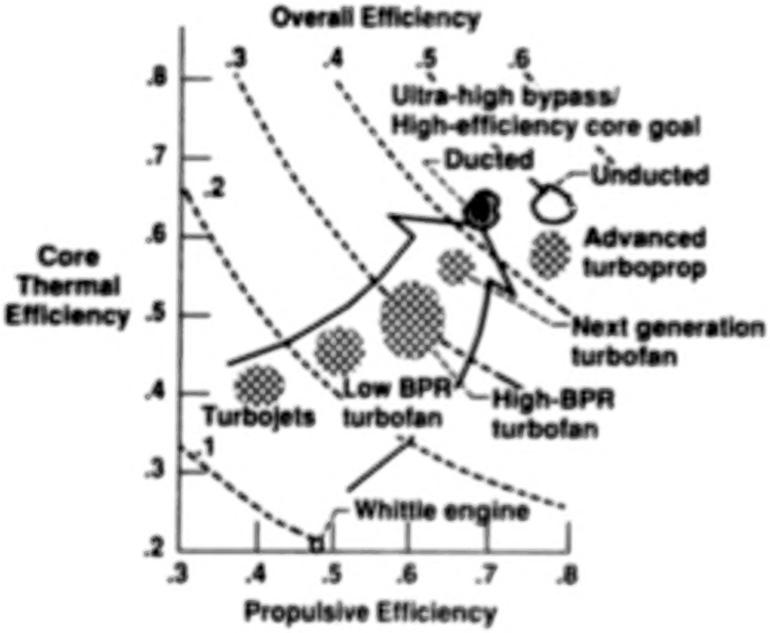
Overview of Subsonic Transport Propulsion Research

Focused Technologies:
- **Ultra-high-bypass ratio cycles**
  - **Ducted props/fans**
  - **Unducted props**
- **High-efficiency core**

CD-91-54500

Propulsion technology for subsonic transports is focused in two main areas: (1) Ultra-high-bypass (UHB) ratio cycles and (2) high-efficiency cores. Previous work demonstrated the technology for fuel-efficient, unducted, advanced turboprops and that effort is concluding. Unducted ultra-high-bypass ratio engines are subject to total thrust limits due to diameter constraints for under-the-wing installations. Thus, current work emphasizes ducted prop/fan configurations suitable for large wide-body aircraft powered by two large-thrust engines mounted under the wing. High-efficiency core investigations center around increasing thermal efficiency by pushing core pressure ratios and temperatures higher. The overall goal is to maximize engine efficiency subject to the environmental constraints of aircraft noise rules and emission limits.

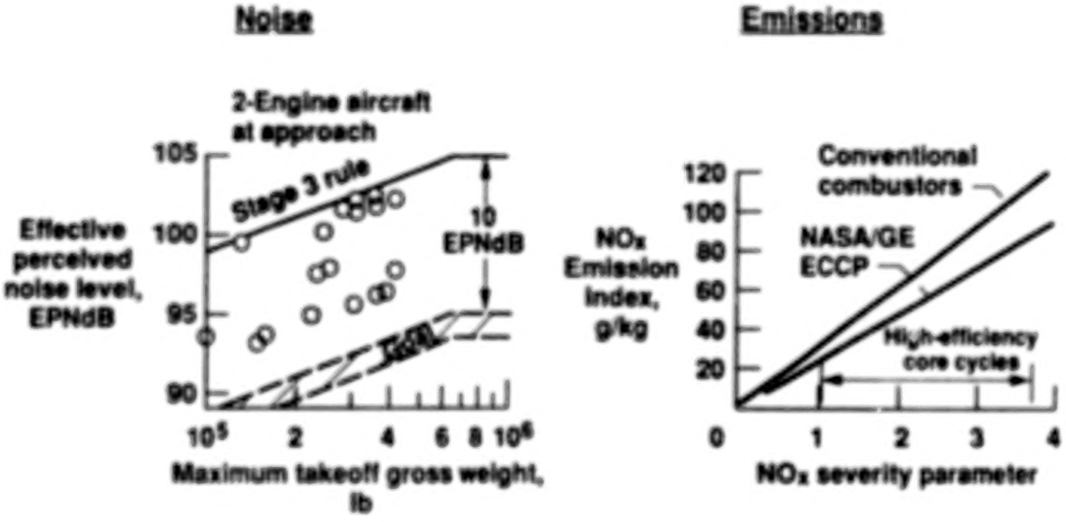
Subsonic Propulsion Efficiencies



CD-91-[illegible]

The contributions of low spool propulsive efficiency and core thermal efficiency to overall propulsion efficiency are shown in this figure. The historic progression is from the first generation turbojets to low-bypass turbofans to current high-bypass turbofans. Advanced turboprop technology represents a major increase in propulsive efficiency. The subsonic goal is to develop the core and low-spool technology to reach the overall efficiency targets shown. In the unducted case, the goal is obtainable through core thermal efficiency gains combined with demonstrated advanced turboprop propulsive efficiencies. However, in the ducted case, new low-pressure spool technology is required in addition to the core improvements.

Subsonic Transport Environmental Constraints



Two environmental constraints which must be satisfied while the efficiency gains are pursued are noise and emissions. Currently, new aircraft must meet the FAR 36, Stage 3 noise rules. The example shown for two-engine aircraft at approach shows that certified effective perceived noise levels (EPNL) for the newest aircraft are already lower than those required for the maximum takeoff gross weights between 100 000 and 400 000 pounds. Part of the reason they are quieter is that the current operating environment is constrained by local airport noise rules which are considerably more stringent than FAR 36, Stage 3. Aircraft noise also poses a significant constraint on the capacity of the air transport system by limiting the hours of operation and airport expansion. Thus, a noise goal for new ultra-high-bypass technology of FAR 36, Stage 3 minus 10 EPNdB has been chosen by NASA for technology development. A band is shown to reflect the fact that a 2 to 3 EPNdB margin must be built in to reduce certification risk. Of course, the goal must be obtained at the takeoff and sideline points, also, and at higher takeoff gross weights.

Emission constraints are not as well defined. Emission index (EI) in grams of NO_x per kilogram of fuel burned is plotted against a NO_x severity parameter which is a function of combustor temperature and pressure. The trends for conventional combustors and the more advanced technology developed under the NASA/GE Experimental Clean Combustor Program (ECCP) are shown. High-efficiency core technology utilizing higher pressures and temperatures will increase the NO_x severity parameter significantly, as indicated by the range labeled on the abscissa for the cycle parameters being studied. Therefore, while specific numerical goals have not been set at present, there is clearly the technical challenge to develop low-NO_x combustors as part of high-efficiency core technology.

Ultra-High Bypass

Unducted props

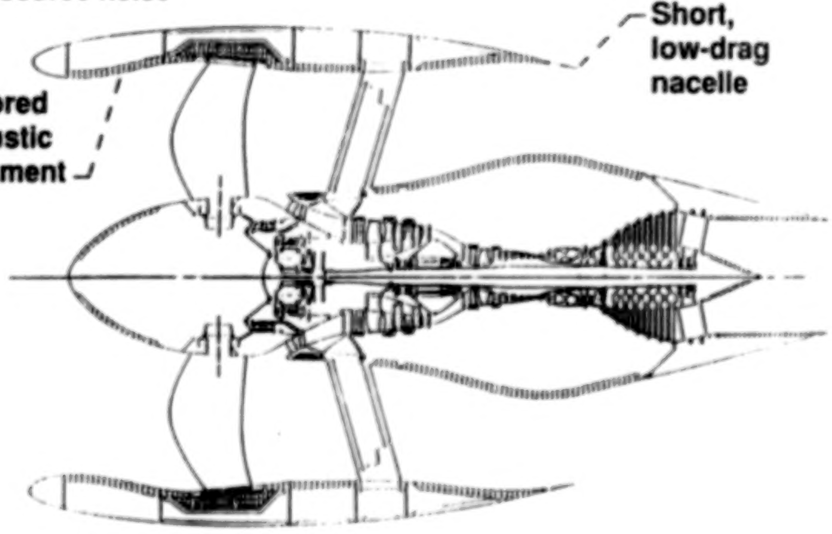
Ducted props/fans

CD-91-54503

The two subdivisions of ultra-high-bypass technology, unducted and ducted props/fans, are shown pictorially as a counterrotation configuration with combined forward and aft swept rotors and a single wide-chord swept rotor in a very short duct. Although the first generation of *aft-swept* advanced turboprop technology is in hand, *forward* sweep is being investigated as part of the conclusion of unducted research. Aerodynamic and acoustic aspects of this work are expected to carry over to a comprehensive investigation of ducted rotors with either forward or aft sweep. The ducted picture comes from computational work to analyze in one integrated computational fluid dynamics (CFD) calculation both the internal and external flow fields. Because the duct is short and thin to minimize weight and cruise drag, not only integrated aerodynamic but also an integrated aeroacoustic analysis will probably be required to capture the performance and noise characteristics of these propulsors.

Ducted Ultra-High Bypass

(Ratios, Bypass 10 to 20+)



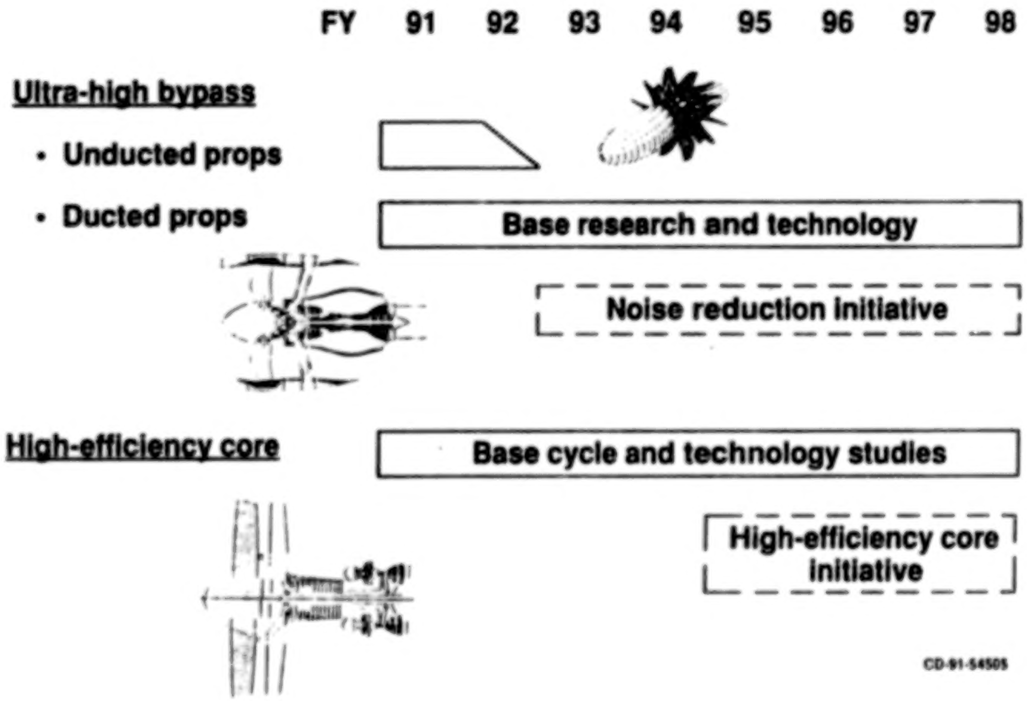
Technical Issues

Geared vs ungeared rotor drives
Variable vs fixed pitch rotors

CD-91-54504

Some features and technology issues associated with ducted ultra-high-bypass propulsors are shown here. The rotor is expected to have fewer blades with wider chords than current turbofans. Bypass ratios greater than 10 and perhaps as high as 20 or more are under consideration. Geared versus ungeared rotor drives and variable-versus fixed-pitch rotors are under consideration. Such choices will be based on the outcome of noise-constrained cycle studies and mission analyses now underway. The short, low-drag nacelle (having a total length perhaps of the order of one rotor diameter) poses particular acoustic problems. Low rotor source noise is a very high priority since space for acoustic treatment is severely limited. Acoustic treatment designs must be highly tailored to and integrated with the prop/fan aeroacoustic design. Rotor stall margin and distortion tolerance must be addressed for prop/fan design parameters and off-design operating conditions not experienced by current turbofans.

Subsonic Transport Aeropropulsion Program



FY 91 92 93 94 95 96 97 98

Ultra-high bypass

- Unducted props
- Ducted props

Base research and technology

Noise reduction initiative

High-efficiency core

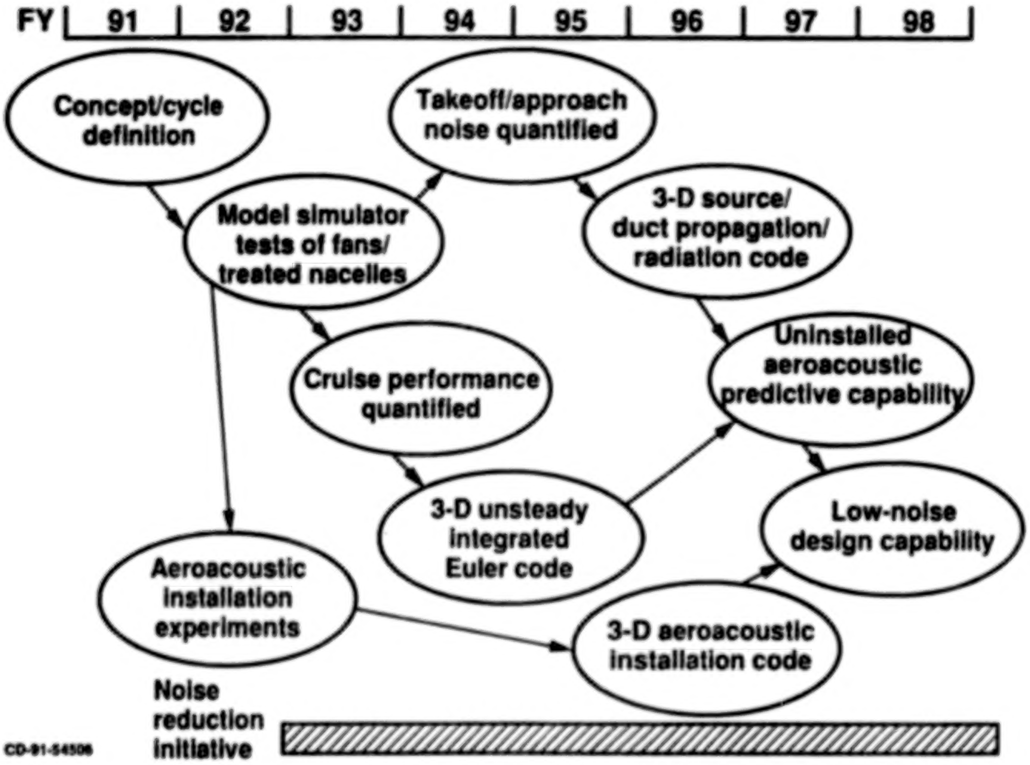
Base cycle and technology studies

High-efficiency core initiative

CD-91-54505

The elements of the overall NASA program in aeropropulsion research for subsonic transports are shown along with the planned sequence. Unducted ultra-high-bypass work is winding down with the effort expected to be essentially complete by the end of FY '92. On the other hand, the base research and technology program in ducted props/fans is well established and growing with a multiyear effort planned. In addition, a noise reduction initiative has been planned for a possible FY '93 start. That effort would draw on the base aerodynamic and source noise reduction technology to arrive at a total design-for-noise capability. An experimental validation using a large powered transport model incorporating high lift and installation aerodynamics technologies with the source noise reduction element would be the program endpoint. Parallel to the ultra-high-bypass propulsor technology is the high-efficiency core research effort. A base program is underway with cycle and technology definition studies nearing completion in preparation for a sustained base technology effort. The intent is to support a high-efficiency core initiative in the FY '95 time period.

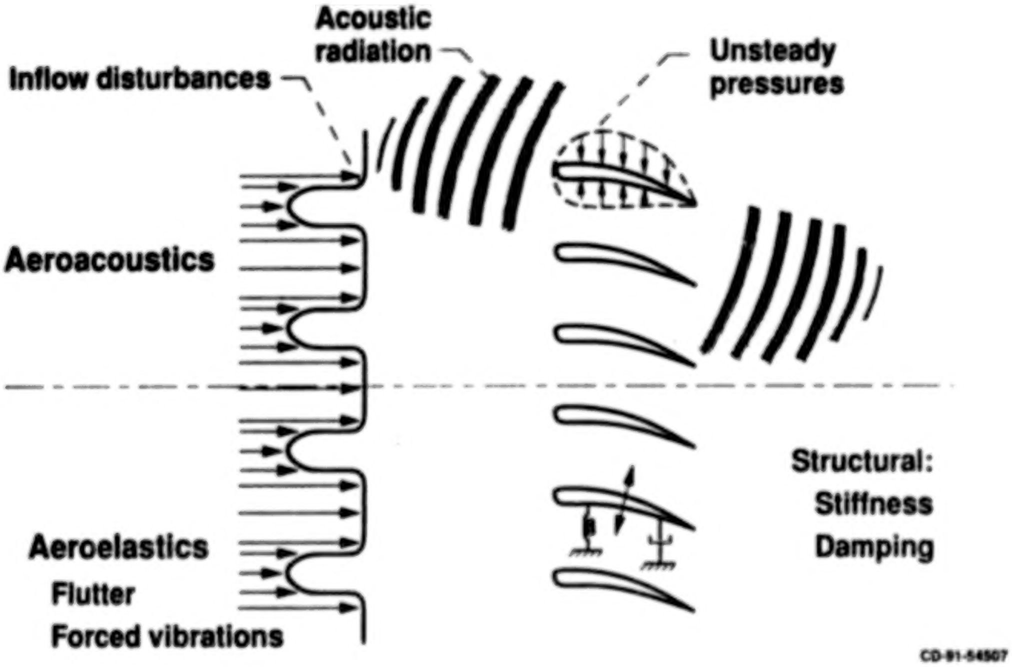
Ducted Prop/Fan Base Research Program


FY
91
92
93
94
95
96
97
98
Concept/cycle definition
Takeoff/approach noise quantified
Model simulator tests of fans/ treated nacelles
3-D source/ duct propagation/ radiation code
Cruise performance quantified
Uninstalled aeroacoustic predictive capability
3-D unsteady integrated Euler code
Low-noise design capability
Aeroacoustic installation experiments
3-D aeroacoustic installation code
Noise reduction initiative

CD-91-54506

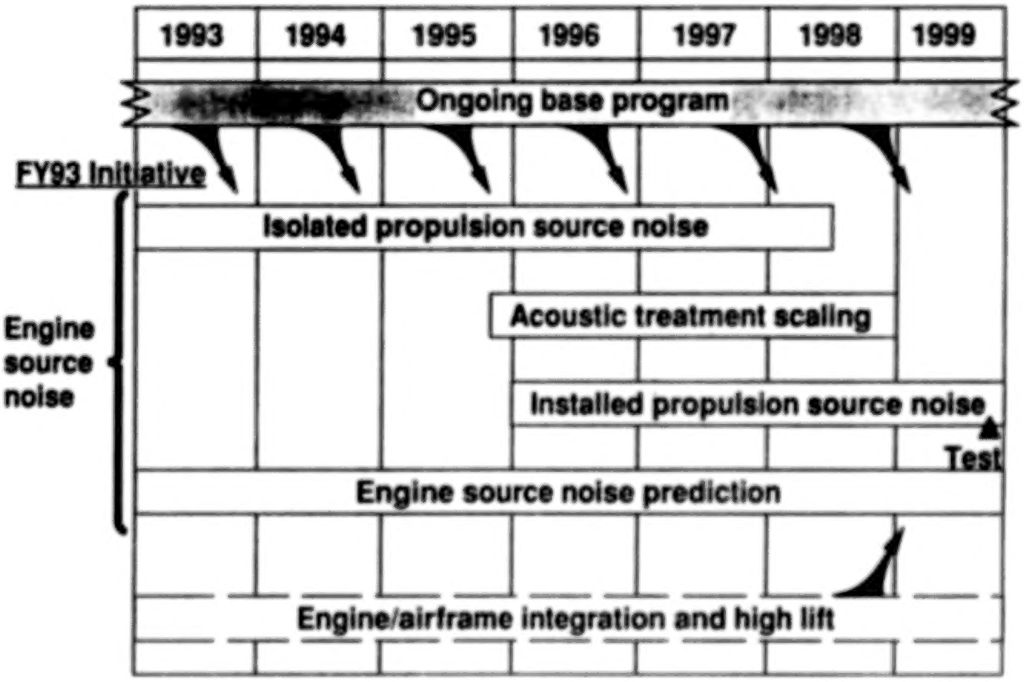
The relations between the main functional elements of the ducted prop/fan research program are shown along with the rough time sequence of the activity. Concept and cycle definition studies are translated into model simulator hardware for wind tunnel tests. Low speed tests quantify takeoff and approach noise and off-design aerodynamic performance. Three-dimensional codes for noise source/propagation/radiation predictions are validated using the wind tunnel data. High-speed tests quantify cruise aerodynamic performance and furnish the data to validate three-dimensional steady and unsteady integrated Euler and Navier-Stokes codes. The low- and high-speed work comes together to provide an uninstalled aeroacoustic prediction capability. Another parallel path of research activity addresses aeroacoustic installation effects both experimentally and computationally. When the uninstalled and installed results are brought together, the ultimate goal of a low-noise, high-efficiency design capability for ducted ultra-high-bypass propulsors is realized. In actuality, the process just outlined, with the exception of cycle definition, may be iterated several times during the 8-year period shown. This base technology will support the planned Advanced Subsonic Transport Noise Reduction Initiative which is planned for FY '93.

Unsteady Aerodynamics Research for Subsonic Propulsion


Acoustic radiation
Inflow disturbances
Unsteady pressures
Aeroacoustics
Structural:
Stiffness
Damping
Aeroelastics
Flutter
Forced vibrations
CD-91-54507

Unsteady aerodynamics is a key area of research encompassing both the aeroacoustic and aeroelastic elements of subsonic propulsion technology. On the aeroacoustic side, inflow disturbances interacting with blade rows produce unsteady blade pressures, which couple with the duct geometry to determine the sound radiated. On the aeroelastic side, flow disturbances drive blade-forced vibrations or couple with blade motions to produce flutter depending on the stiffness and damping characteristics of the blade row. Thus, the understanding and modeling of the unsteady interactions of blade rows with flows is essential to the ability to design for low noise and structural integrity. For these reasons the ducted prop/fan base research program contains considerable work on blade row unsteady aerodynamics.

Advanced Subsonic Noise Reduction Initiative

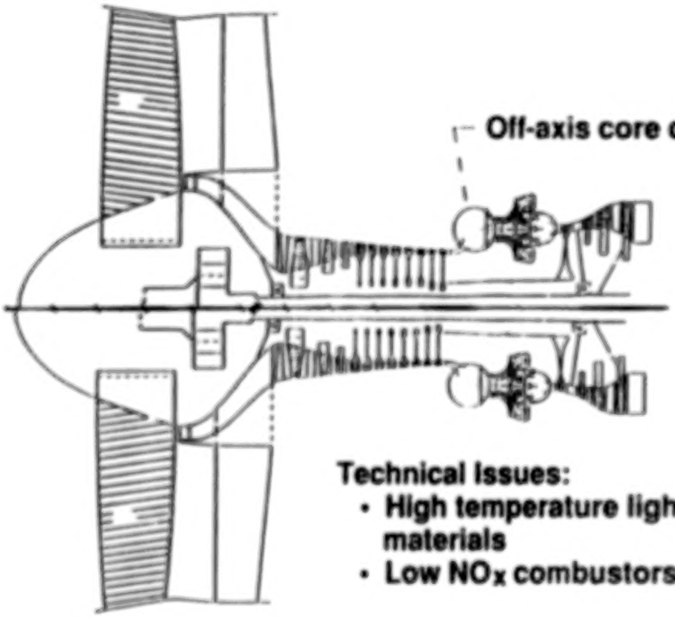

1993
1994
1995
1996
1997
1998
1999
Ongoing base program
FY93 Initiative
Isolated propulsion source noise
Engine source noise
Acoustic treatment scaling
Installed propulsion source noise
Test
Engine source noise prediction
Engine/airframe integration and high lift

CD-91-54508

Elements of the engine source noise portion of the planned FY '93 Advanced Subsonic Noise Reduction Initiative are shown. Isolated source noise reduction work will focus on experiments using integrated fan/treated nacelle simulators. Acoustic treatment scaling work will establish scaling methodology by a combination of simulator and full-scale engine experiments. Installed propulsion source noise efforts will culminate in the aeroacoustic test of a large airplane model with propulsion simulators in the NASA Ames 40 x 80 Wind Tunnel. Accompanying these three efforts is the development and validation of engine source noise prediction methodology subject to the constraints of efficiency and structural integrity. Two other elements of the total noise reduction initiative planned (engine/airframe integration and high lift systems) determine the configuration of the airplane model and the way the simulators are installed on it. The ongoing base program supplies technical concepts and tools for use in the highly focused noise reduction initiative.

High-Efficiency Core

(Overall pressure ratio, 50 to 100; combustor inlet temperature, >1000 °F)



Technical Issues:
- High temperature light-weight materials
- Low NO_x combustors

CD-91-54509

Some features and technology issues associated with high-thermal-efficiency cores are shown. Overall pressure ratios being investigated range from 50 to 100, and combustor inlet temperatures are greater than 1000 °F. High-temperature, lightweight materials with minimal cooling requirements are needed along with combustor designs to limit NO_x, in spite of the high temperatures and pressures. In such a compression system at high pressure ratios, conventional compressor design approaches lead to very small flow path and blade heights. Consequently, the usual problems in attaining high component efficiencies in small turbomachinery are multiplied, and unconventional geometries such as off-axis core modules are being studied. For example, each module might contain a centrifugal compressor, combustor, and radial turbine; and there may be several of these modules spaced circumferentially around the engine axis.

CONCLUDING REMARKS

In the subsonic propulsion research area, noise reduction is emerging as a dominant theme as a means to alleviate noise constraints on the capacity of the air transport system. The impact is greatest on the design of the low-pressure (prop/fan) portion of the engine because the dominant noise source is the rotor. Since a large body of technology was developed for unducted rotors over the last decade and application of that technology to commercial products has been delayed by market forces, current research emphasis is on technology for ducted configurations suitable for new twin-engined, long-range aircraft. A parallel effort to develop high thermal efficiency cores is about to move into base technology work now that cycle studies have identified the technology issues and concepts to address them.

REFERENCES

1. Groeneweg, J.F.: Aeroacoustics of Advanced Propellers. NASA TM-103137, 1990.

2. Groeneweg, J.F.; and Rice, E.J.: Aircraft Turbofan Noise. J. Turbomachinery, vol. 109, no. 1, Jan. 1987, pp. 130-141.

3. Groeneweg, J.F.; and Bober, L.J.: NASA Advanced Propeller Research. NASA TM-101361, 1988.

4. Groeneweg, J.F., et al.: Turbomachinery Noise. AEROACOUSTICS OF FLIGHT VEHICLES, THEORY, AND PRACTICE, H.H. Hubbard, ed., (to be published).

N91-20117

ULTRA-HIGH BYPASS RESEARCH

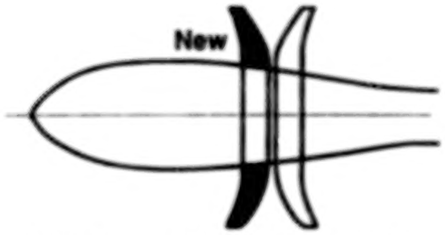
Christopher J. Miller, James H. Dittmar, and Robert J. Jeracki
NASA Lewis Research Center
Cleveland, Ohio

This paper presents an overview of the ultra-high bypass research at the NASA Lewis Research Center. Over the past decade, the program was focused on unducted rotors and precipitated several development efforts in industry. In this area, forward-swept unducted rotors are the concluding effort. Forward sweep can reduce the tip vortex strength and has a potential for reducing the noise of counter-rotating unducted rotors. The aerodynamic performance can also be improved slightly over an aft-swept blade.

The future direction of the program is toward ultra-high bypass ratio ducted machines, and short cowls as a particular item. Short cowls can have less aerodynamic drag and less weight, but may allow more noise to be radiated. In general, the aerodynamic research effort will be to provide higher efficiencies, sufficient flow conditioning, and attached flow at high angles of attack. Acoustically, the research is directed towards developing an understanding of fan acoustics in short ducts. The reduced duct length means that there might be insufficient duct length for acoustic cutoff. With less length and less cowl thickness, the space for acoustic treatment is limited, requiring integrated aeroacoustic designs.

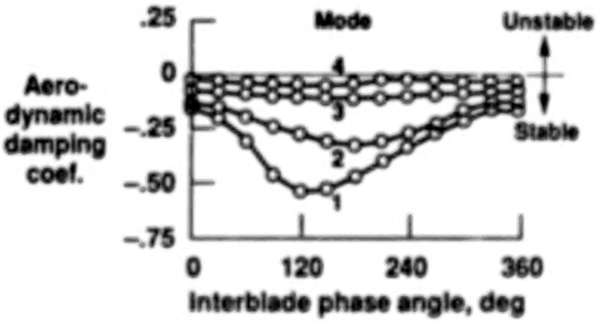
Advanced Unducted Aerodynamics

Forward-Swept Counterrotation Concept

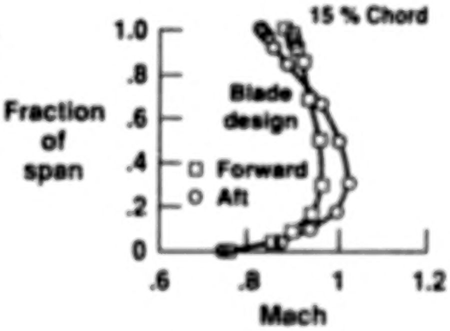


- Lower peak Mach: higher efficiency
- Weaker tip vortex: lower noise
- Aeromechanically stable

New forward-swept forward row



Aeromechanical stability



Peak through-flow Mach

CD-91-54472

The current program in advanced unducted propfan concepts is concluding with a study of the effect of forward sweep on noise and aerodynamic performance. A counterrotation configuration is used with new forward-swept front blades and existing aft-swept rear blades. The new blades are designed to have the same loading as the original aft-swept front blades.

Aerodynamically, several effects from forward sweep are seen. Forward sweep places the blade tip in a region of slightly lower axial velocity which reduces the section Mach numbers and also evens out the radial distribution of peak through-flow Mach number, resulting in higher efficiency. The tip vortex development is strongly effected by sweep. An aft-swept blade, particularly with the leading edge vortex (LEV) present at high-lift/off-design conditions, has a very strong tip vortex. Significant amounts of low-energy fluid are collected off the blade and produce a large low speed core. In comparison, the forward-swept blade will have separate leading edge and tip vortices with the boundary layer fluid primarily in the LEV.

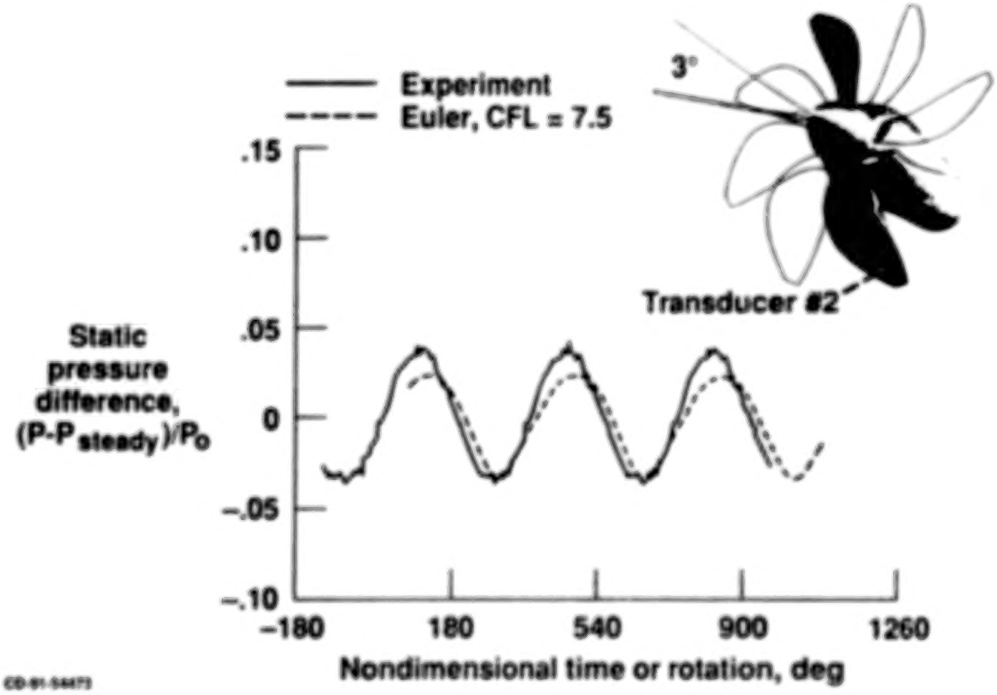
A strong noise source is the front-row tip vortex interacting with the aft-blade row. This acoustic source is dependent on the vortex strength and is stronger with increasing radius. Here, the forward-swept blade provides specific benefits: the tip vortex is much weaker, and the separate LEV, when present, will interact with the aft blades well inboard of the tip. The increased spacing between the blade rows at the tip should decrease noise, whereas the proximity of the blades near the hub increases the potential field interaction noise.

Aeromechanically, the forward-swept blade is stable throughout the operating range.

Unsteady Aerodynamics

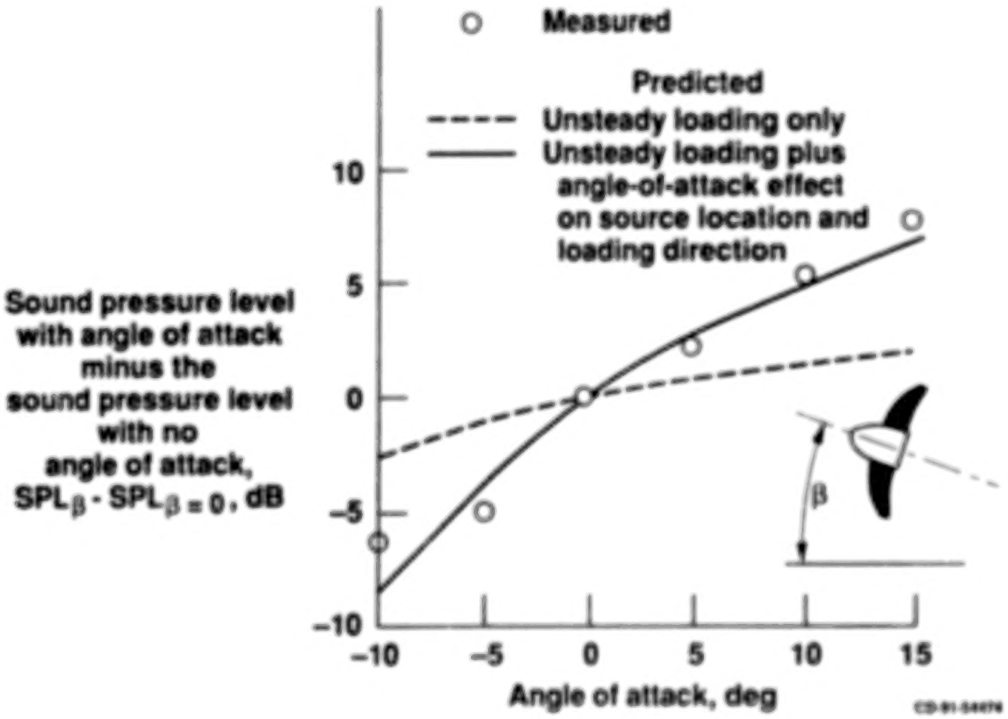
SR7 2-Bladed Propfan Modane Test Case (M = 0.8, J = 3.06, AOA = 5°)

Unsteady surface pressure (fine grid)



Unsteady blade surface pressures are needed to predict angle-of-attack performance and as input for acoustic calculations. Several computer codes now provide very good predictions of these pressures. For example, a full-scale, single-rotation SR-7 propfan was tested at Modane, France to provide unsteady blade surface pressures. For the test point shown, a highly loaded climb-out condition, model drive power constraints prevented reaching the correct blade loads with more than two of the eight blades. Euler calculations by Nallasamy (ref. 1), and by the Allison Gas Turbine Division of General Motors Corp. (shown) provide very good agreement with the experimental data.

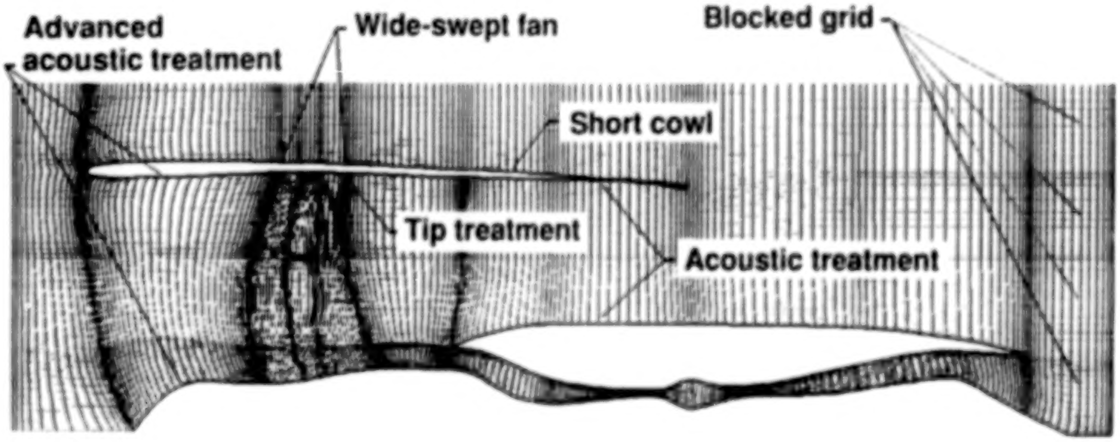
Angle-of-Attack Noise Prediction



When provided with good unsteady loading information, noise prediction techniques can now accurately predict the unsteady noise of a propfan at an angle of attack. Mani (ref. 2) developed an analysis that predicted the change in noise as a function of angle of attack due to both the steady and unsteady loading. The analysis includes the azimuthal variation of the radiation efficiency of the steady noise sources. This analysis, however, was limited to small angles of attack. Krejsa (ref. 3) extended the work to be valid at large angles of attack. When results from Krejsa's analysis are compared with the data of Woodward (ref. 4), the significant improvement in prediction accuracy is evident.

Ultra-High Bypass Directions

- Develop high-efficiency propulsors
- Lower noise than current technology
- Higher fan pressure ratios
- Lower fan tip speeds

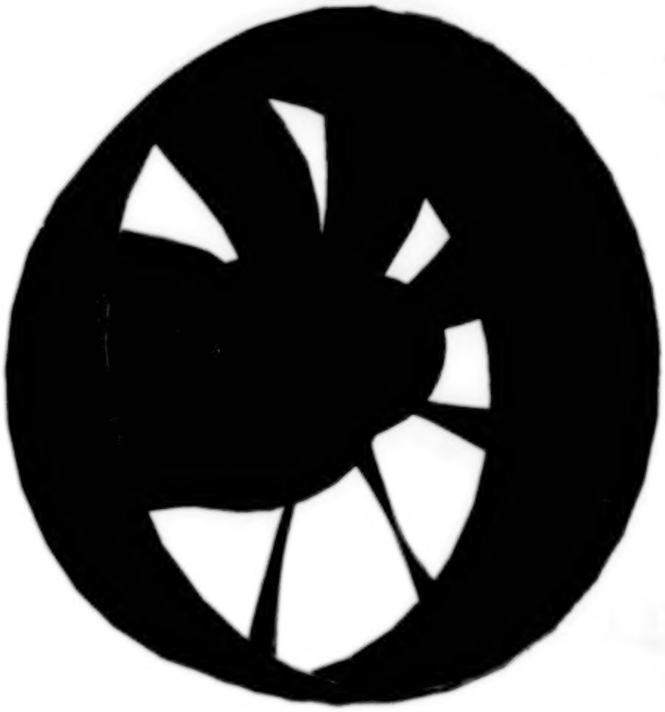


CD-91-54475

Advanced unducted propfans show significant performance gains over older unducted technology, but diameter constraints and the need for higher thrust levels have shifted NASA's research to ducted configurations again. The goal now is to obtain higher efficiencies and lower noise than current technology by using the lessons learned with unducted concepts. To address higher thrusts with constrained diameters, higher fan pressure ratios are a priority, but they incur an efficiency and noise penalty. Current aerodynamic research focuses on fans with a lower number of wide-chord swept blades, significantly shorter cowls, and tip treatments. Lower fan tip speeds are seen as the primary driver to reduce source noise. Acoustic research concepts include advanced acoustic treatments, nonaxisymmetric arrangements, and active controls.

To evaluate these new concepts, NASA is developing computational fluid dynamics (CFD) codes for ducted high-bypass fan sections under contract to the Allison Gas Turbine Division of General Motors Corp. (ref. 5). This group of Advanced Ducted Propfan Analysis Codes (ADPAC) handle steady and unsteady inflows with the Euler and Navier-Stokes equations in average-passage or time-accurate formulations. Currently under development is a code using a blocked grid that will solve for the internal (fan flow), external, and core flow about a realistic engine configuration - the NASA E^3 engine. A parallel effort is underway within NASA to incorporate acoustic models for noise generation and propagation so that one code will provide fluid mechanical and aeroacoustic evaluation of these concepts.

Ducted Propfan at Angle of Attack



CD-91-54476

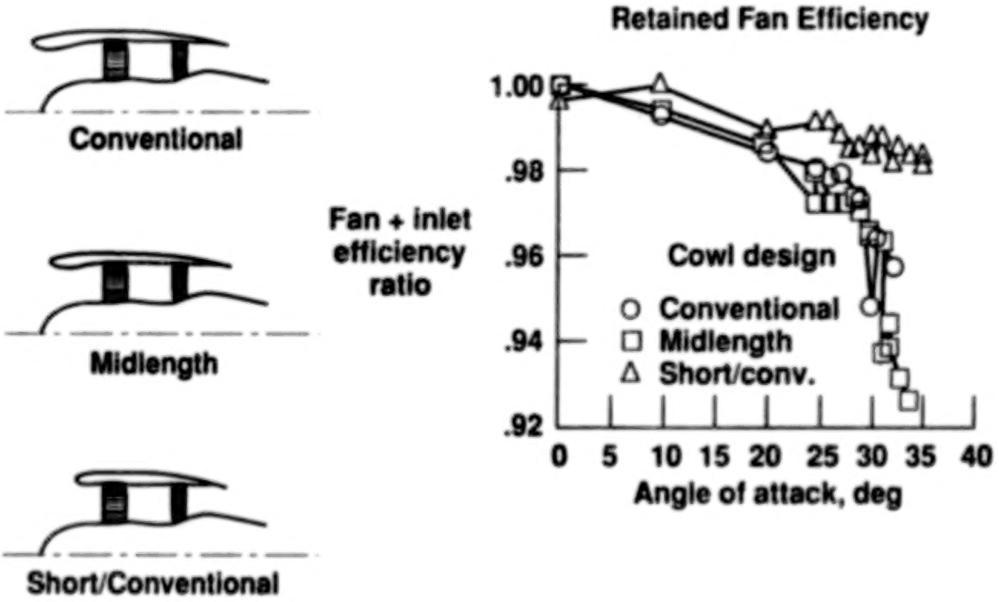
A demonstration geometry for the ADPAC codes was created by taking the NASA single-rotation SR7 blade geometry, clipping it at the 75-percent span, and placing it in a representative short cowl. There is a tip gap of about 2-percent span. The flow conditions are Mach 0.8, advance ratio of 3.12, and axis of rotation at 5° to the inflow.

The Euler calculation used a somewhat coarse mesh with 20 points per chord line. The flow solution required approximately three revolutions to reach a consistent time history and took 20 hr of Cray Y-MP time. The surface static pressure shows the asymmetric loading due to angle of attack.

Because of the long times required for fine-mesh calculations, the development of the codes will be directed towards parallel processing.

Advanced Ducted Propeller Test Results

Angle of Attack Effects by Cowl Length

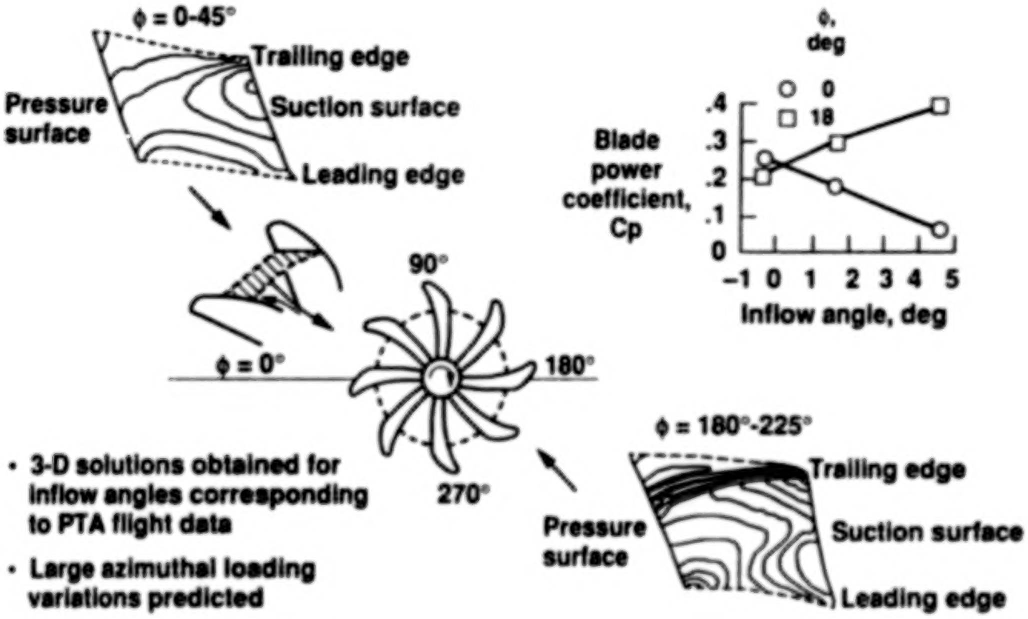


CD-91-54477

Under terms of a space act agreement, Pratt & Whitney Aircraft and NASA are testing an Advanced Ducted Propeller (ADP) concept model to evaluate ducted propfan performance with shorter, low-drag cowl designs. The results show that very short cowls have desirable aerodynamic features. Shown is the combined fan plus inlet efficiency ratioed to the value at zero angle of attack. Although the long cowl can do a better job of reducing distortion, it will also develop more of a boundary layer. Also, at the high angles of attack, the short cowl appears to have less separated flow coming into the fan. The preliminary data show that the combined short-length cowl/ conventional spinner + fan performs well at high angle of attack: the short cowl and fan have the same efficiency reduction at 35° as the conventional and midlength cowls have at 20°. It is not clear whether this high angle-of-attack performance is due to the reduced cowl separation or to retained fan efficiency.

CFD Calculations

Unsteady Euler Analysis of the Flow Field of a Propfan at an Angle of Attack


ϕ = 0-45°
Trailing edge
Pressure surface
Suction surface
Leading edge
90°
ϕ = 0°
180°
270°
ϕ = 180°-225°
Trailing edge
Pressure surface
Suction surface
Leading edge
ϕ, deg
○ 0
□ 18
Blade power coefficient, Cp
.4
.3
.2
.1
0
−1
0
1
2
3
4
5
Inflow angle, deg

- 3-D solutions obtained for inflow angles corresponding to PTA flight data
- Large azimuthal loading variations predicted
- Passage shocks form and dissolve during one revolution

Pressure Contours
r/R = .66; $\alpha = 4.6°$

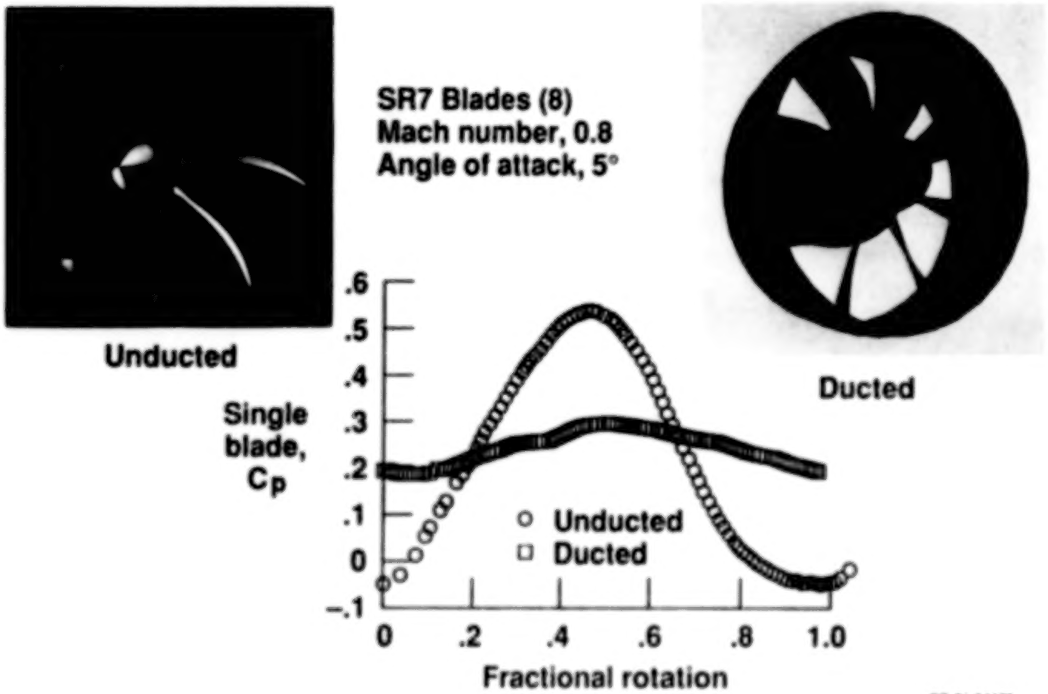
CD-91-54479

Angle-of-attack Euler code predictions provide insight into experimental results obtained in the Propfan Test Assessment (PTA) flight test program (ref. 6). The Euler predictions show large azimuthal loading variations, a strong driver in noise generation. The strongest driver is the unsteady nature of the passage shock. On the highly loaded downward portion of the revolution, a strong passage shock forms and is attached to the trailing edge on the suction surface of the blade. This is the portion of the blade rotation where the generated noise is directed downwards towards the "community." On the upward moving side, the passage shock dissolves, reducing the noise directed towards the cabin. The predictions show that at cruise Mach numbers relatively small inflow angles can produce very large blade-loading variations. If the flow were straightened, as by a short cowl, the unsteady loading and, thus, the noise would be significantly reduced at nominal angle-of-attack conditions.

These loadings have been used as input to an acoustic code to predict near field noise levels (ref. 7). Qualitatively, the effect of inflow angle is correctly predicted.

Unsteady Aerodynamics

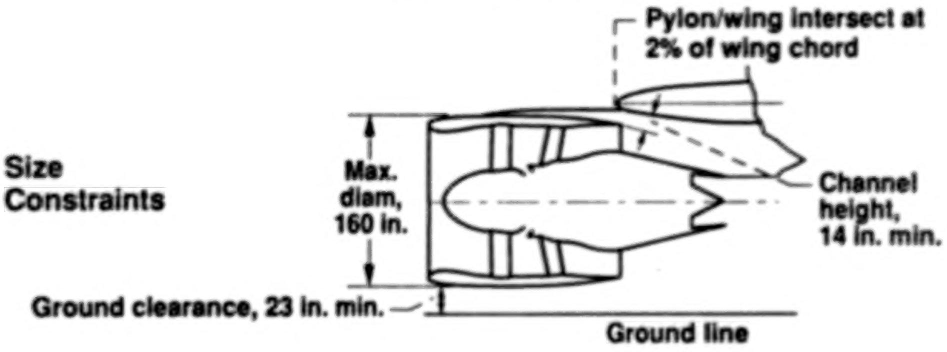
Duct Reduces Unsteady Loading



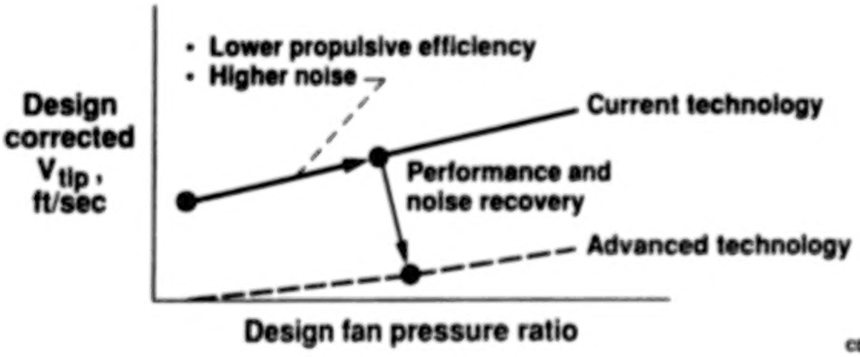
CD-91-54479

The flow-straightening effect of a short cowl is demonstrated by a side-by-side comparison of similar propfans. The unducted propfan is the SR7 with eight blades operating at Mach 0.8 and an angle of attack of 5°. This is a relatively high-loading condition. The ducted propfan is based on the SR7 blade, clipped at 75-percent span, and operating at the same conditions as the unducted SR7. The plot of normalized-blade power loading shows that the cyclic variation of loading is about five times larger for the unducted propfan.

New Cycles


Size Constraints
Pylon/wing intersect at 2% of wing chord
Max. diam, 160 in.
Channel height, 14 in. min.
Ground clearance, 23 in. min.
Ground line

Impact on Fan Tip Speed and Pressure Ratio

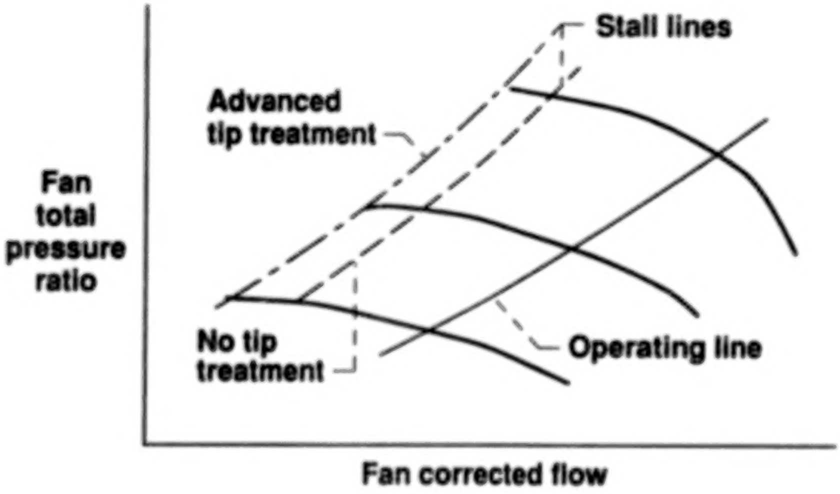


CD-91-54480

The need for larger thrust engines for twin-engine, long-range aircraft is constrained by engine diameter. Further, to reduce the fan noise level, one of the most effective techniques is the reduction in fan tip speed. Designers are therefore presented with the task of reducing fan tip speed while increasing fan pressure ratio. Shown is a result from a Pratt & Whitney cycle study to identify current directions and critical technology items. Note that the desired change in fan pressure ratio and fan tip speed is a significant change from current technology. To operate at these highly loaded conditions, fan surge and stall margins are even more important.

Casing Treatments

Fan Showing Improved Stall Margin

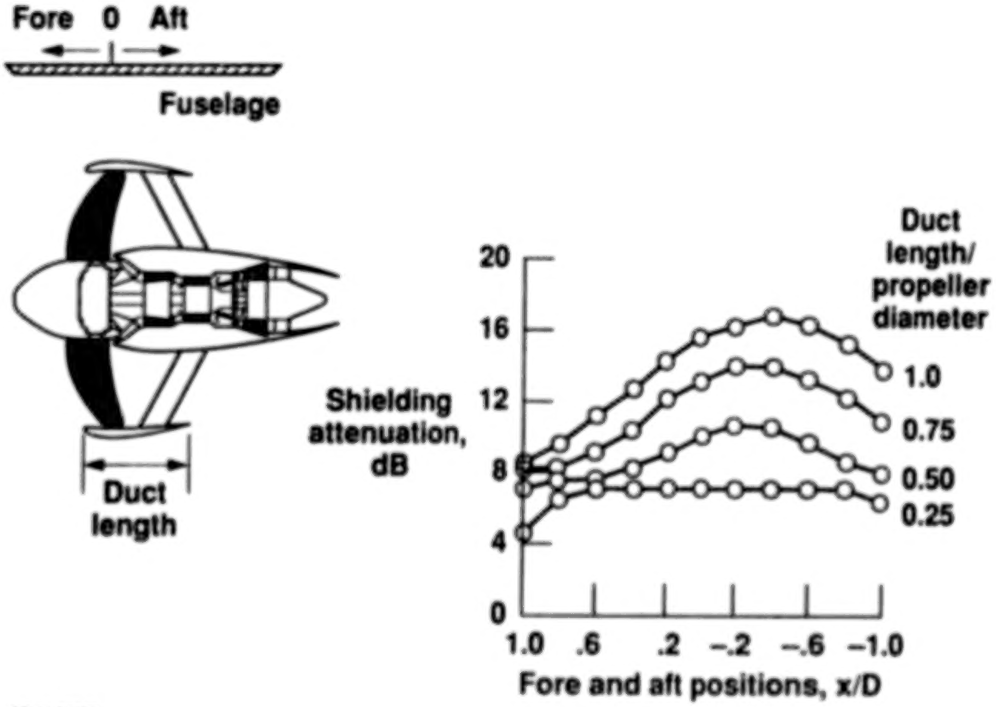


CD-91-54481

Higher fan pressure ratios and lower fan tip speeds increase the blade loading everywhere, especially at the tip. Advanced casing treatments can improve the surge margin and stall performance by delaying separation and stall at the tip. The Pratt & Whitney cycle study evaluated the performance of an advanced tip/casing treatment. This treatment provides not only a significant improvement in surge margin but has almost no aerodynamic efficiency penalty. Stall margin can be a problem with current technology since the stall margin is decreased as the fan loading is increased. By using tip treatments, though, current stall margins can be maintained at the higher loadings needed for future engines.

On design, tip treatments must have low losses and may reduce noise by reducing the strength of or promoting the dissipation of the tip vortex. Weaker vortices reduce the unsteady loading on stators/EGV's, which can reduce noise.

Predicted Short-Duct Noise Shielding



CD-91-54482

The acoustic shielding provided by a cowl has a large effect on cabin noise. To evaluate short-duct acoustic shielding for a wing-mounted engine, a simple barrier-shielding model was used to estimate the noise on the fuselage (ref. 8). With the propeller located one-third of the duct length from the inlet, estimates for the maximum blade passing tone attenuation varied from 7 dB for a duct length of 0.25 D (propeller diameter, D) to 16.75 dB for a duct length of 1 D. Attenuations for the higher harmonics would be even larger because of their shorter wavelengths relative to the duct length. These estimates show that the fuselage noise-reduction potential of a ducted, as compared with an unducted, propeller is significant even for very short duct lengths.

Summary

- **Unducted rotor research concluding**
- **Emphasis shifting to ducted configurations**
- **Initial short-duct aerodynamic results encouraging**
- **Developing aeroacoustic understanding of props and fans in short ducts**
- **CFD analysis tool development emphasizing integrated calculations for rotors in short cowls**

CD-91-54483

Forward sweep can reduce the tip vortex strength and, hence, has a potential for reducing the noise for counter-rotating unducted rotors. The aerodynamic performance can also be improved slightly over an aft-swept blade.

Short cowls have most of the aerodynamic advantages of conventional cowls with very few disadvantages. Experiments show that they do a good job of flow straightening and have delayed lip separation as compared with conventional length cowls. The reduced length also means less boundary layer buildup, less weight, and less drag.

NASA's acoustic research effort is currently directed towards developing an understanding of propeller/fan acoustics in short ducts. The reduced duct length means that there might be insufficient duct length for acoustic cutoff. With less length and less cowl thickness, the space for acoustic treatment is limited, requiring integrated aeroacoustic designs.

Continuing CFD analysis tool development will provide Euler and Navier-Stokes codes for advanced high-bypass ratio engine concepts. These tools, which do an integrated calculation of the rotor and cowl flow fields, will handle steady inflow as well as angle-of-attack calculations.

REFERENCES

1. Nallasamy, M.; and Groeneweg, J.F.: Unsteady Blade-Surface Pressures on a Large-Scale Advanced Propeller: Prediction and Data. NASA TM-103218, 1990.

2. Mani, D.: The Radiation of Sound from a Propeller at Angle of Attack. NASA CR-4264, 1990.

3. Krejsa, E.A.: Prediction of the Noise from a Propeller at Angle of Attack. NASA TM-103627, 1990.

4. Woodward, R.P.: Measured Noise of a Scale Model High Speed Propeller at Simulated Takeoff/Approach Conditions. NASA TM-88920, 1987.

5. Hall, E.J.; Delaney, R.A.; and Bettner, J.L.: Investigation of Advanced Counterrotation Blade Configuration Concepts for High Speed Turboprop Systems, Task 1: Ducted Propfan Analysis. NASA CR-185217, 1990.

6. Nallasamy, M.; and Groeneweg, J.F.: Unsteady Euler Analysis of the Flowfield of a Propfan at an Angle of Attack. AIAA Paper 90-0339, Jan. 1990.

7. Nallasamy, M., et al.: Near-Field Noise of a Single Rotation Propfan at an Angle of Attack. AIAA Paper 90-3953, Oct. 1990 (also, NASA TM-103645, 1990).

8. Dittmar, J.H.: An Estimate of the Noise Shielding on the Fuselage Resulting from Installing a Short Duct Around an Advanced Propeller. NASA TM-100262, 1988.

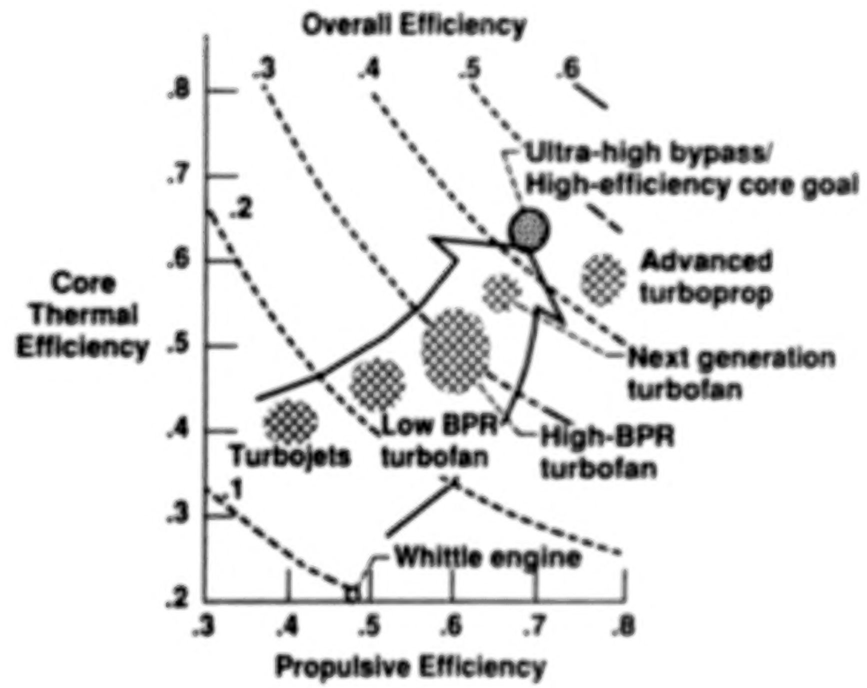
N91-20118

HIGH-EFFICIENCY CORE TECHNOLOGY

Robert M. Plencner and Gerald Knip, Jr.
NASA Lewis Research Center
Cleveland, Ohio

Studies were undertaken to determine the potential benefits from implementing aircraft turbine engine core technology well beyond that being developed for the next generation of long-haul subsonic transport engines (i.e., entry into service date of 1993). These core improvements, projected for year 2010 technology, include the use of very high-pressure-ratio cycles, advanced lightweight materials with minimal cooling requirements, and improved component efficiencies. The studies indicate that a large improvement is possible with engines using these advanced cores as compared to the current turbine engine designs. This paper reports on the results of the studies and identifies the key challenges to achieving the predicted improvements in performance.

Civil Engine Efficiency Trends



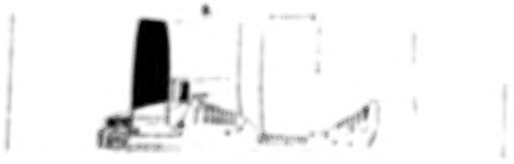
CD-91-54182

Large gains in turbine engine overall efficiency have been made since the first turbojets were introduced. Recent advances such as high-bypass turbofans and the advanced turboprop have resulted largely in improvements in propulsive efficiency. The goal of the current set of studies was to emphasize the core in order to investigate the potential of improving the thermal efficiency over that of the next generation turbofan. A shift in the efficiency trend is shown by the arrow in the figure.

High-Efficiency Core Study

Objective: **assess benefits of advanced high-efficiency cores for large and small aircraft turbine engines and identify key technologies**

Engine



Scope:
- **OPR 50 - 100**
- **FPR 1.3 - 1.5**
- **2 & 3 Spools**
- **Direct & geared fans**
- **In-line & off-axis cores**
- **Year 2010 Tech. readiness**

Contractors:
- **General Electric**
- **Pratt & Whitney**
- **Allison**

Mission

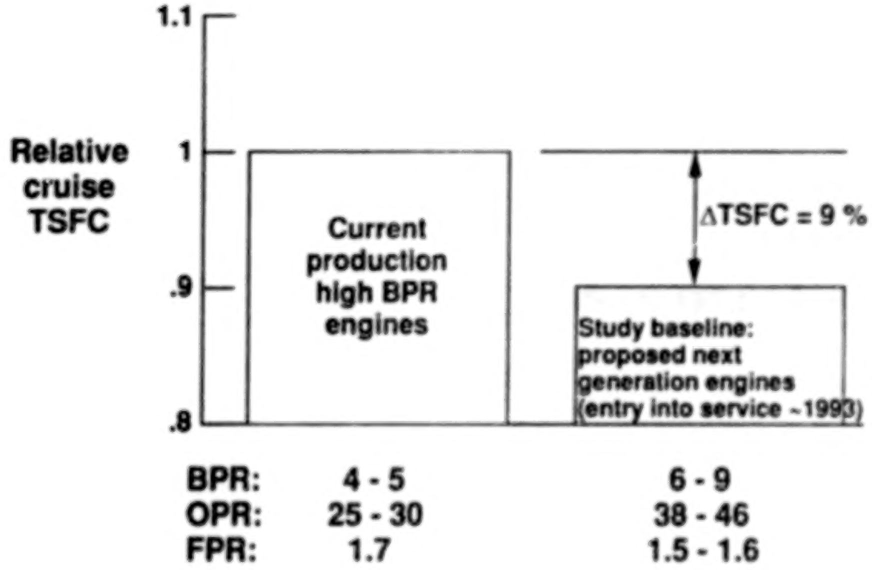
4-engine aircraft
Speed, Mach 0.85
Range, 5000 nm
Passengers, 500

Assessment criteria:
- **Fuel burned**
- **DOC**

CD-91-54183

Studies were undertaken to assess the potential benefits of advanced high-efficiency cores, to develop a preliminary mechanical design for assessment of concept feasibility, and to determine the key technologies required for implementation. Engines with overall pressure ratios (OPR's) of 50 to 100, fan pressure ratios (FPR's) of 1.3 to 1.5, and uncooled or minimally cooled hot sections were evaluated. Direct and geared fans were considered as well as in-line and off-axis core configurations. In all cases, the bypass ratios were optimized to take advantage of the high specific power cores which result from the high pressure and temperature conditions. Year 2010 technology readiness was assumed for all engine technologies. General Electric and Pratt & Whitney assessed large engines of about 50 000 pounds of thrust suitable for a 500-passenger advanced subsonic aircraft. The advanced engines were evaluated in terms of both fuel burned and direct operating cost (DOC) improvements for this aircraft relative to the baseline engine. In addition, Allison studied an advanced, high-efficiency, 8000-shaft horsepower (SHP) turboshaft engine suitable for high-speed rotorcraft. Only engine performance was evaluated in this study.

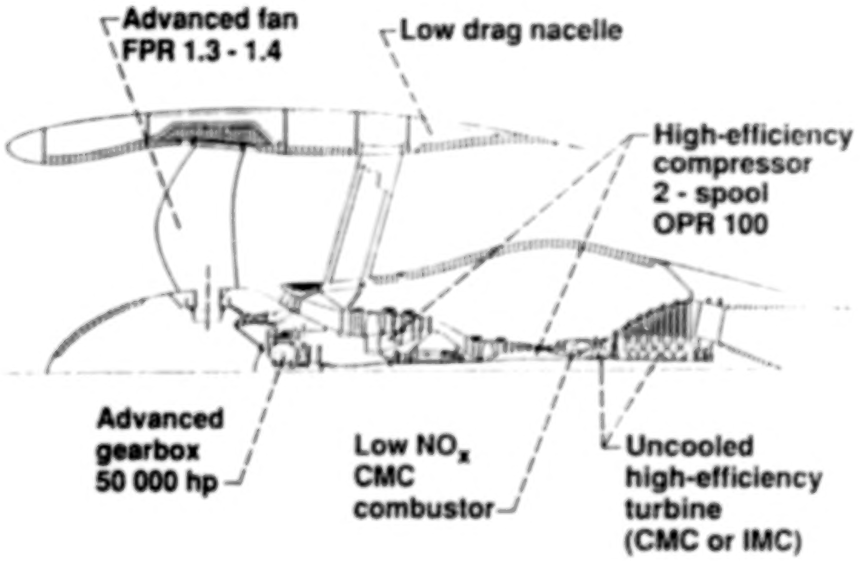
Subsonic Transport Baseline Engine Definition



CD-91-54184

The baseline engine chosen for the large-engine studies was a proposed next-generation turbofan engine with an entry into service date of 1993. This baseline engine has approximately 9 percent lower cruise thrust specific fuel consumption (TSFC) than current production high-bypass-ratio (BPR) engines. A comparison of cycle performance for the baseline engine and the current production engine is shown above.

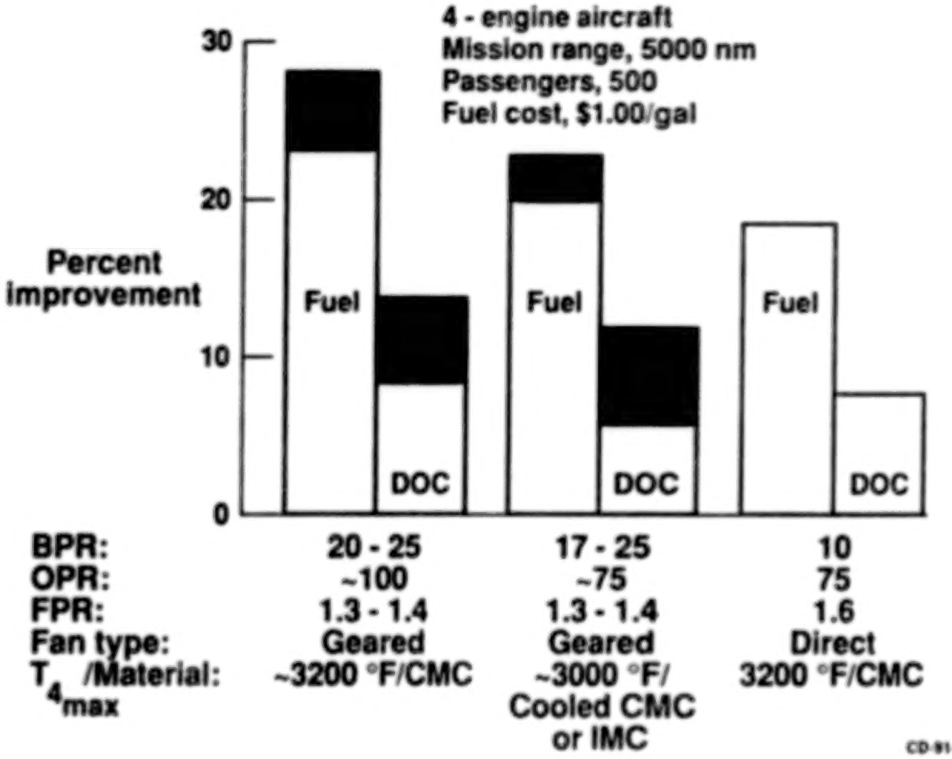
In-Line Engine Configuration


Advanced fan
FPR 1.3 - 1.4
Low drag nacelle
High-efficiency
compressor
2 - spool
OPR 100
Advanced
gearbox
50 000 hp
Low NO_x
CMC
combustor
Uncooled
high-efficiency
turbine
(CMC or IMC)

CD-91-54185

The typical 100:1 overall pressure ratio engine that resulted from the studies consists of a two-spool geared configuration with a bypass ratio of 20 to 25. The resulting fan pressure ratios are 1.3 to 1.4. Low drag nacelles are required to minimize the losses associated with the high bypass ratios. Efficiency improvements are needed in both the compressor and the turbine to enable thermal efficiency improvements at the higher pressure ratio. Advanced materials such as ceramic matrix composites (CMC) and intermetallic matrix composites (IMC) are used extensively throughout the hot section of the engine to reduce or eliminate cooling flow requirements. Since the fan must be geared to achieve the very high bypass ratios, advanced gearbox technology will be needed to achieve the required transmission power of about 50 000 hp. As a result of the high overall pressure ratio, the combustor entrance pressure and temperature are very high. This would result in NO_x formations exceeding current levels with current technology combustors. Therefore, NO_x combustor technology must be developed for very small combustors in this type of engine.

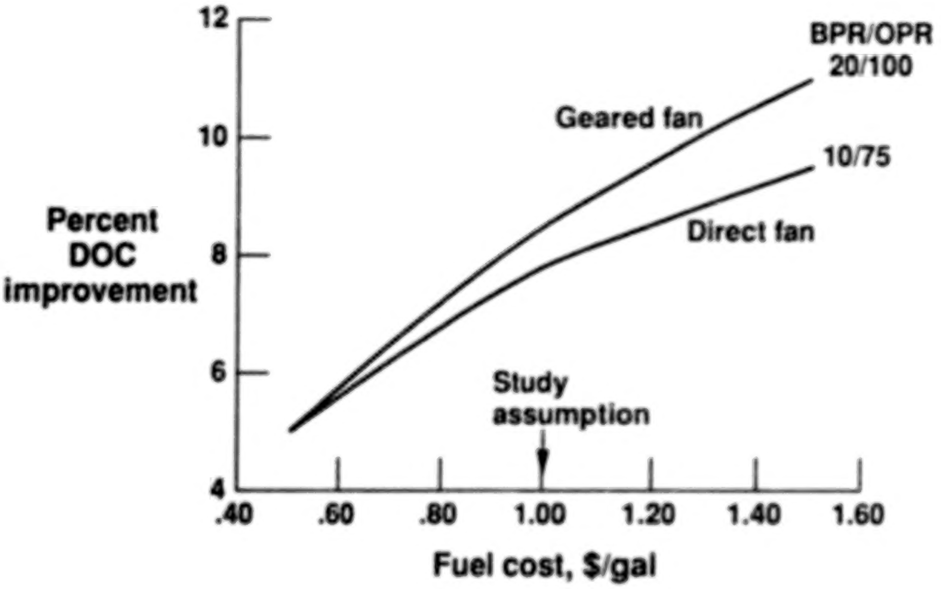
Subsonic Transport Engine Results



CD-91-54186

Reductions in fuel usage of 19 to 28 percent and in direct operating cost of 6 to 14 percent were projected for the advanced study engines. The shaded bands represent a range of cycle and technology assumptions. A geared engine with a very high bypass ratio of 20 to 25, overall pressure ratio near 100, and a turbine inlet temperature of 3200 °F resulted in a fuel savings of approximately 25 percent and a DOC improvement of nearly 10 percent. Limiting the hot section temperature to 3000 °F (assuming a less aggressive turbine material technology), resulted in a fuel savings greater than 20 percent and a DOC improvement near 10 percent. Finally, with a direct (non-geared) fan, the bypass ratio limited to 10, and an overall pressure ratio of 75, fuel savings were near 20 percent and the DOC improvement approached 10 percent.

Impact Of Fuel Cost On DOC

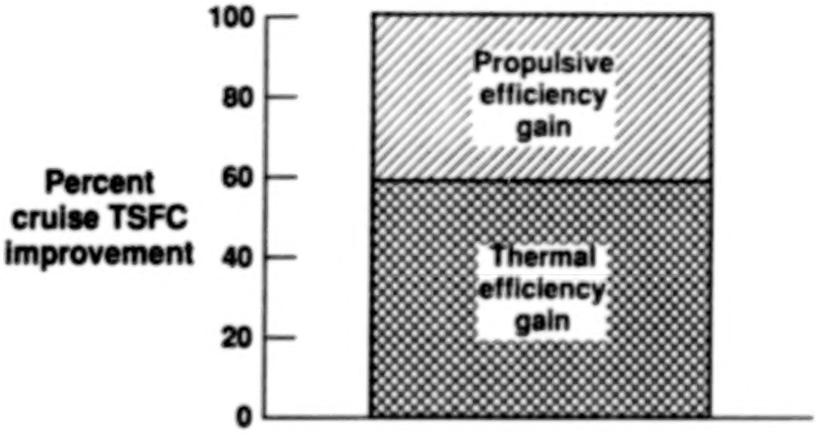


CD-91-54187

World fuel prices are very volatile and will likely remain so. The cost of fuel can have a dramatic effect on the direct operating cost realized with high-efficiency cores. For example, a threefold increase in fuel cost yields a twofold improvement in DOC. This is a result of the fuel price becoming a larger percentage of DOC.

The future fuel cost may also have an effect on the engine configuration that is most cost effective. At low fuel costs there is little or no benefit for a geared fan engine in terms of DOC. The higher initial cost and higher maintenance cost outweigh the lower fuel consumption. However, at the study assumption of $1.00/gal, there is a small benefit shown for the geared fan configuration. This benefit continues to increase as the fuel costs increase. Thus, fuel prices well above $1.00/gal may require the development of the technologies associated with geared fan engines.

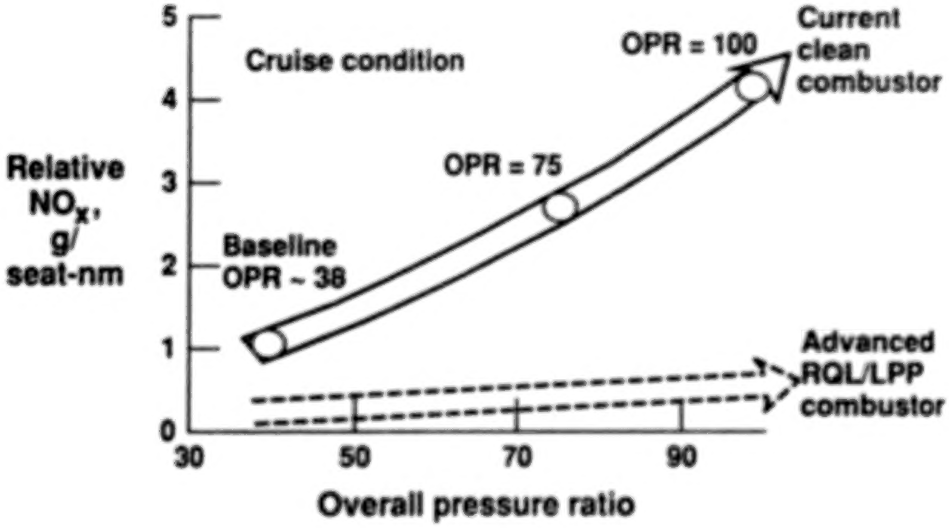
Performance Improvement Breakdown



CD-91-541[illegible]

The combination of increased pressure ratio, improved core component efficiencies, and advanced materials that allow higher turbine temperature with reduced cooling flow directly provide a thermal efficiency gain that accounts for about 60 percent of the overall performance improvement. These same advanced core materials and improved core components provide the increased core specific power that enables a large part of the propulsive efficiency gain through the use of higher bypass ratios. Low drag, thin nacelles, and improved low-spool component efficiencies provide the remainder of the propulsive efficiency gain.

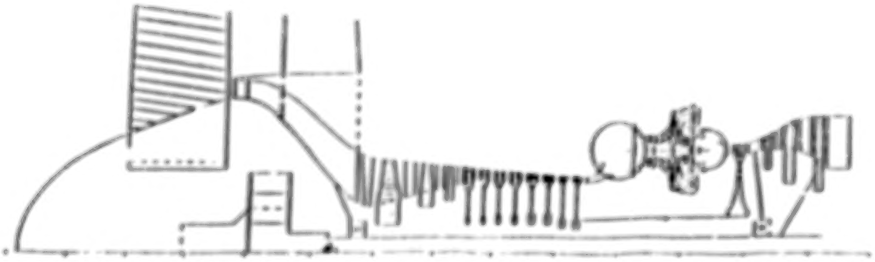
Engine Emissions Challenge



CD-91-54189

One of the challenges resulting from the use of the high-pressure-ratio cores is a reduction of NO$_x$ emissions. As the pressure ratio is increased, the combustor entrance pressure and temperature increase. The severe combustor entrance conditions produce very high NO$_x$ emissions compared to the baseline configuration, even when fuel savings are included. These high emissions can potentially be reduced to levels at or below those found in the baseline through the use of rich-burn/quick-quench/lean-burn (RQL) or lean-premixed-prevaporized (LPP) combustors which are being developed under the High-Speed Research (HSR) program. However, the combustor required for the high-pressure-ratio subsonic engine is an order of magnitude smaller in airflow size compared to the HSR combustor. Therefore, additional work may be required to develop the RQL and LPP technology for such small combustors.

Unconventional Core Engine



Potential Benefits of Off-Axis Core

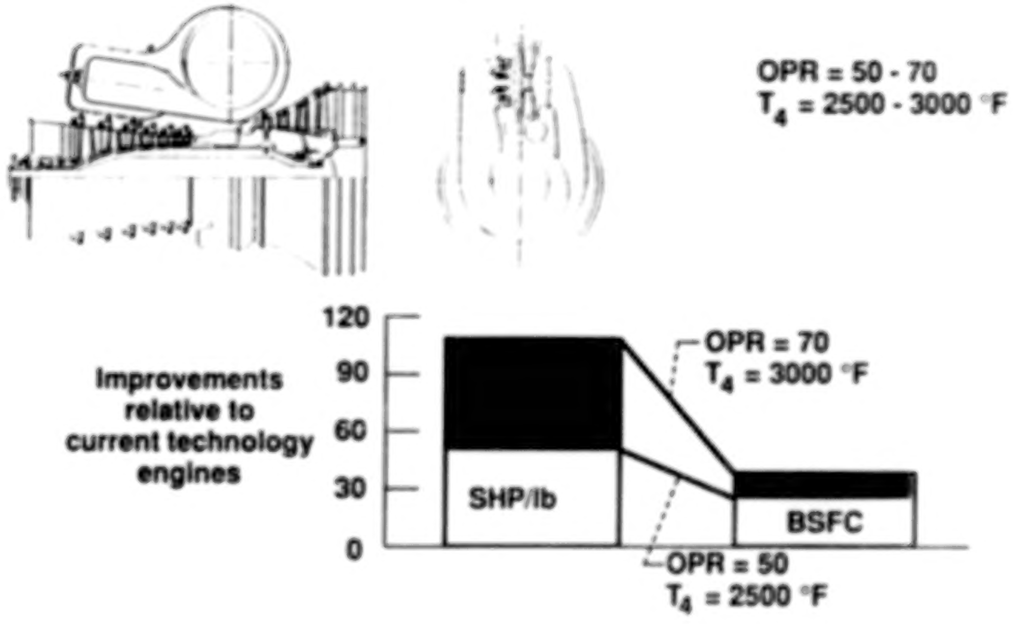
- **Reduced component losses**
- **Lower part count**
- **Core reliability**
- **Ease of maintenance**

CD-91-54190

A conventional in-line engine with a very high pressure ratio has small compressor stages leading to potentially significant aerodynamic losses and mechanical design problems. An alternative under consideration is to duct the flow to one or more cores that are off-axis from the main engine. Because an off-axis core does not have a fan or low-spool shaft running through it, the hub diameter can be reduced. This adds design flexibility to the high pressure spool so many of the aerodynamic and structural penalties from small sizes in an in-line configuration can be avoided. Also, since centrifugal compressor and radial in-flow turbine stages are feasible in an off-axis configuration, the parts count can be reduced compared to a conventional all axial configuration. If multiple off-axis cores are used, there is redundancy in the hot section which could potentially increase reliability. Hot section maintenance could also be simplified because the low pressure spools would not have to be removed for access to the high pressure spool. Such an off-axis arrangement is being evaluated as part of a follow-on study.

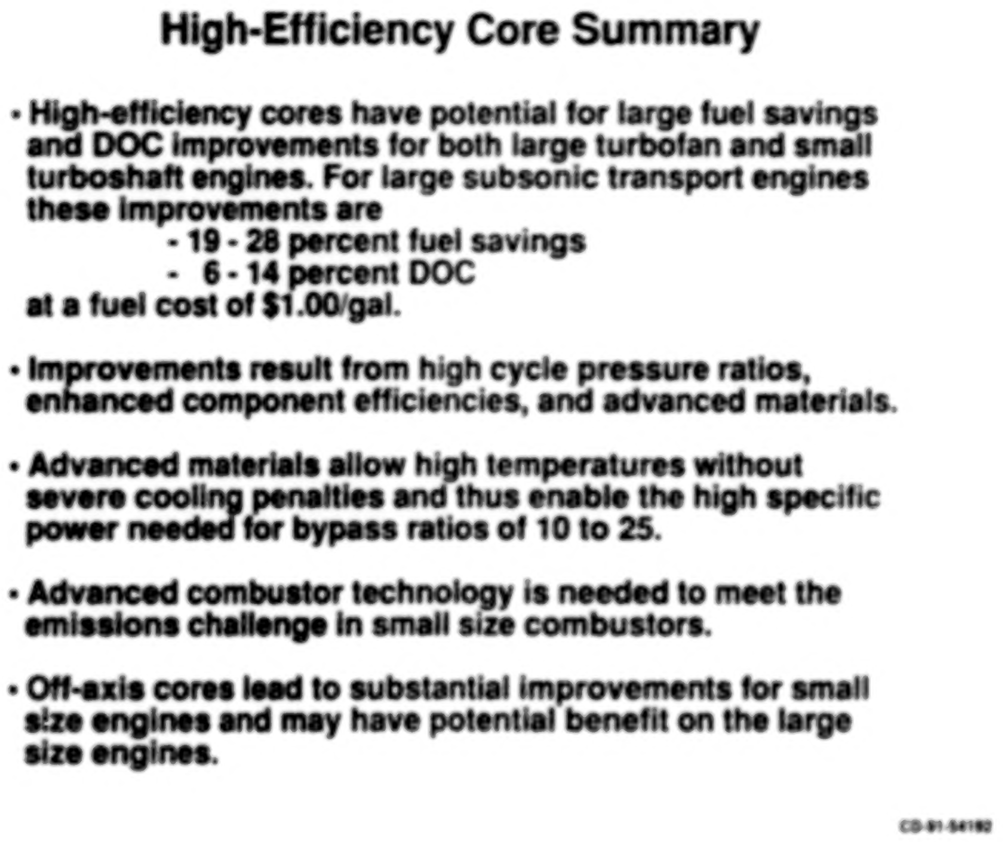
Small Engine Study

8000 SHP



CD-91-54191

Allison undertook a study to assess the benefits of advanced high-efficiency cores for an 8000-SHP turboshaft engine relative to a current technology engine. All performance improvements were evaluated in terms of engine parameters. Because of the small size of the engine in this study, an off-axis core configuration was the only viable design to result from the investigation. The design arrangement had the core axis 90 degrees to the main axis of the engine. Two levels of technology were assumed. The first design assumed a hot section material temperature limit of 3000 °F, which resulted in an optimum overall pressure ratio of 70. The second design assumed materials that were only capable of 2500 °F, which resulted in an overall pressure ratio of 50. The two levels of technology are shown as bands on the figure. The study projected a potential brake specific fuel consumption (BSFC) reduction of more than 30 percent with a specific weight improvement of 50 to 100 percent.



High-Efficiency Core Summary

- High-efficiency cores have potential for large fuel savings and DOC improvements for both large turbofan and small turboshaft engines. For large subsonic transport engines these improvements are
  - 19 - 28 percent fuel savings
  - 6 - 14 percent DOC

  at a fuel cost of $1.00/gal.
- Improvements result from high cycle pressure ratios, enhanced component efficiencies, and advanced materials.
- Advanced materials allow high temperatures without severe cooling penalties and thus enable the high specific power needed for bypass ratios of 10 to 25.
- Advanced combustor technology is needed to meet the emissions challenge in small size combustors.
- Off-axis cores lead to substantial improvements for small size engines and may have potential benefit on the large size engines.

CD-91-54192

The advanced high-efficiency core study engines with an entry into service date of 2015 have the potential for large fuel savings and DOC improvements for both large turbofan and small turboshaft engines. The fuel savings resulting from the large subsonic transport engines were 19 to 28 percent when compared to a proposed next-generation turbofan engine with an entry into service date of 1993. The corresponding DOC improvements are 6 to 14 percent for a fuel cost of $1.00/gal. These improvements result from high cycle pressure ratios, high bypass ratios, enhanced component efficiencies, and advanced materials. The advanced materials allow high temperatures without severe cooling penalties and thus enable the high specific power needed for bypass ratios of 10 to 25.

The small component sizes associated with high pressure engines create new challenges, such as controlling emissions and minimizing aerodynamic losses in turbomachinery. Advanced combustor technology is needed to meet the emissions challenge. Also, unconventional off-axis cores, which lead to substantial improvements for small size engines, may also have potential benefit on the large size engines. Studies are currently underway to investigate the potential benefits of off-axis cores in large engines.

N91-20119

MULTIDISCIPLINARY RESEARCH OVERVIEW

(IHPTET/NPSS)

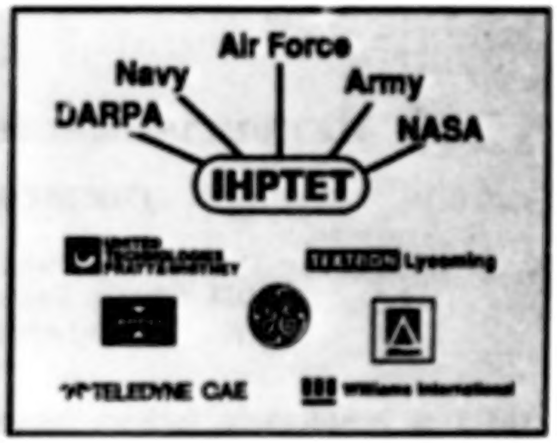
Lester D. Nichols and Susan M. Johnson
NASA Lewis Research Center
Cleveland, Ohio

The Integrated High Performance Turbine Engine Technology (IHPTET) Program and the Numerical Propulsion System Simulation (NPSS) Program are two aeropropulsion multidisciplinary efforts at NASA Lewis that complement each other. The IHPTET initiative is an experimental program to advance engine development and double propulsion system capability by the turn of the century. NASA Lewis is contributing, with the Department of Defense and seven major aerospace contractors, to the development of these advanced, military, high-performance engines in the areas of compressors, combustors, turbines, nozzles, controls, mechanical systems, instrumentation, materials, structures, and computational fluid dynamics.

The NPSS effort is a computational, long-range program with the goal of reducing the cost and time of developing advanced-technology propulsion systems. This goal will be achieved through a cooperative effort of NASA, industry, universities, and other Government agencies to develop the necessary technologies for integrating disciplines, components, and high-performance computing into a user-friendly simulation environment. This simulation will allow for comprehensive evaluation of new concepts early in the design phase, before a commitment to hardware is made. It will also allow for rapid assessment of field-related problems, particularly where operational problems are encountered during conditions that are difficult to simulate experimentally. Data generated from the IHPTET engines will be used to help validate NPSS computations.

IHPTET Team

INTEGRATED HIGH PERFORMANCE TURBINE ENGINE TECHNOLOGY
AIR FORCE
NAVY
DARPA
INDUSTRY
NASA
ARMY


Air Force
Navy
Army
DARPA
NASA
IHPTET
United Technologies Pratt & Whitney
Textron Lycoming
Teledyne CAE
Williams International

The IHPTET initiative represents a revolutionary new approach to engine development in which advanced materials, innovative structural designs, improved aerothermodynamics and advanced computational methods are synergistically combined to double propulsion system capability by the turn of the century.

CD-91-54129

NASA Lewis is one of the team members involved in the Integrated High Performance Turbine Engine Technology (IHPTET) Program. The Department of Defense (DOD) and seven major aerospace contractors are also team members in this program to double turbine engine propulsion system capability by the turn of the century. NASA Lewis is contributing IHPTET-related research and technology to the program in terms of component research (i.e., compressors, combustors, turbines, mechanical systems, nozzles, and controls) and critical disciplines (e.g., materials, structures, instrumentation, and computational fluid dynamics).

NASA's IHPTET-Related R&T

- Emphasis on long-term, fundamental R&T
  - Strengthens fundamental knowledge base for IHPTET
  - Affects later phases of IHPTET

- Significant effort in numerical modeling of propulsion aerothermodynamics, structures, controls, etc.
  - Validated codes for analysis and design components
  - High potential for integrated disciplines

- Significant effort in advanced composite materials
  - Emphasis on improved fibers, processing methods, and design and life-prediction methods

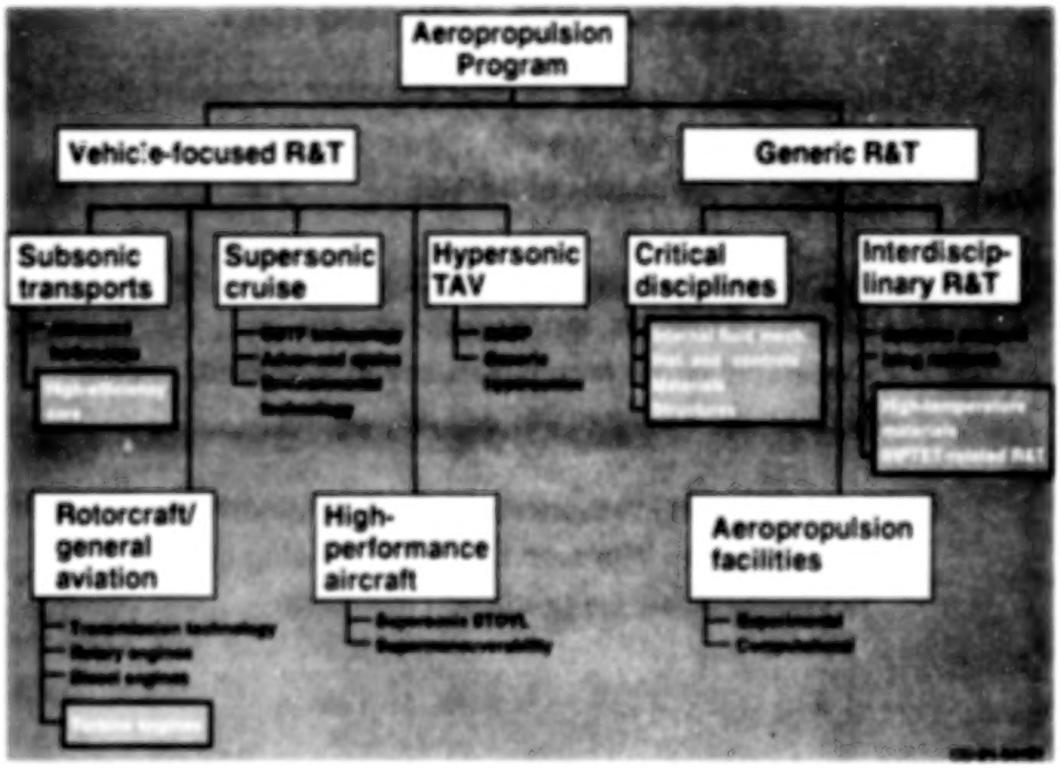
CD-91-54130

NASA Lewis' contributions to the IHPTET program are long-term, fundamental research and technology. Lewis' research efforts explore and strengthen the knowledge base of the IHPTET program. Results of this research will affect the later phases of the IHPTET program.

Lewis has a significant effort in the numerical modeling of propulsion aerothermodynamics, structures, controls, etc. Lewis leads the computational fluid dynamics effort in the IHPTET program.

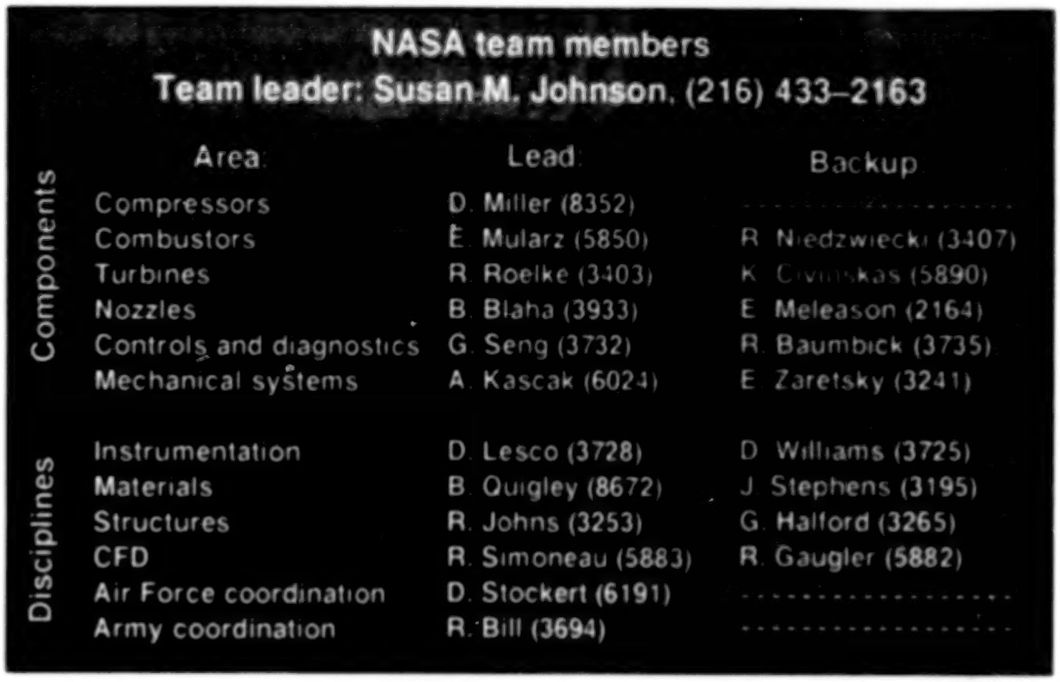
The advanced composite materials effort at Lewis is contributing to the IHPTET program through the Lewis HITEMP program. Intermetallic matrix composites and ceramic matrix composites are being researched, and the results are incorporated into the IHPTET program. Polymer matrix composites are also being researched within the controls and diagnostics team for improved fibers and processing methods.

NASA Lewis IHPTET-Related R&T
Represented by Highlighted Blocks


Aeropropulsion Program
Vehicle-focused R&T
Generic R&T
Subsonic transports
Supersonic cruise
Hypersonic TAV
Critical disciplines
Interdisciplinary R&T
Internal fluid mech.
Materials
Structures
High-temperature materials
IHPTET-related R&T
Rotorcraft/ general aviation
High-performance aircraft
Aeropropulsion facilities
Transmission technology
Rotary engines
Diesel engines
Turbine engines
Supersonic STOVL
Supermaneuverability
Experimental
Computational

The IHPTET-related areas of research in the NASA Lewis Aeropropulsion Program are highlighted in this figure. Under vehicle-focused R&T, IHPTET-related research is found in subsonic transports and rotorcraft/general aviation. Under generic R&T, IHPTET-related research is in critical disciplines and interdisciplinary R&T.

IHPTET-Related R&T Planning Team



NASA team members

Team leader: Susan M. Johnson, (216) 433–2163

	Area	Lead	Backup
Components	Compressors	D. Miller (8352)	- - - - - - - - - - - - - - - -
	Combustors	E. Mularz (5850)	R. Niedzwiecki (3407)
	Turbines	R. Roelke (3403)	K. Civinskas (5890)
	Nozzles	B. Blaha (3933)	E. Meleason (2164)
	Controls and diagnostics	G. Seng (3732)	R. Baumbick (3735)
	Mechanical systems	A. Kascak (6024)	E. Zaretsky (3241)
Disciplines	Instrumentation	D. Lesco (3728)	D. Williams (3725)
	Materials	B. Quigley (8672)	J. Stephens (3195)
	Structures	R. Johns (3253)	G. Halford (3265)
	CFD	R. Simoneau (5883)	R. Gaugler (5882)
	Air Force coordination	D. Stockert (6191)	- - - - - - - - - - - - - - - -
	Army coordination	R. Bill (3694)	- - - - - - - - - - - - - - - -

CD-91-84132

This figure presents the NASA Lewis team members for IHPTET-related research. Each member is a focal point in her or his component or discipline area. Team members' telephone numbers, without the (216) 433- prefix, are given after their names.

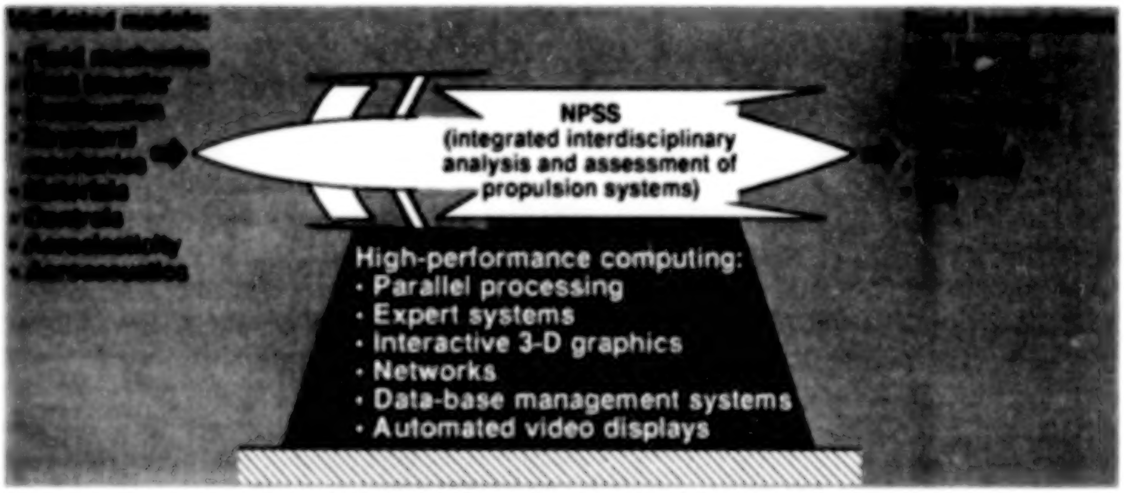
NPSS Goal

- Reduce life-cycle costs by advancing system analysis capability through high-fidelity computational simulations by
  - Higher level of concurrent engineering
  - Rapid evaluation of effects of new and novel concepts on system performance
  - Early risk assessment
  - Early operability studies
  - Rapid evaluation of field problems
  - Assessment of performance degradation

CD-91-54134

Implementing new technology in aerospace propulsion systems is becoming prohibitively expensive. One of the major contributors to the high cost is the need to perform many large-scale system tests. Extensive testing is used to capture the complex interactions among the multiple disciplines and the multiple components inherent in complex systems. The objective of the Numerical Propulsion System Simulation (NPSS) Program is to provide insight into these complex interactions through computational simulations. The tremendous progress taking place in computational engineering and the rapid increase in computing power expected through parallel processing make this concept feasible within the near future. However, it is critical that the framework for such simulations be put in place now to serve as a focal point for the continued developments in computational engineering and computing hardware and software. The NPSS concept will provide that framework.

NPSS—A Numerical Test Cell for Aerospace Propulsion Systems



NPSS
(integrated interdisciplinary analysis and assessment of propulsion systems)

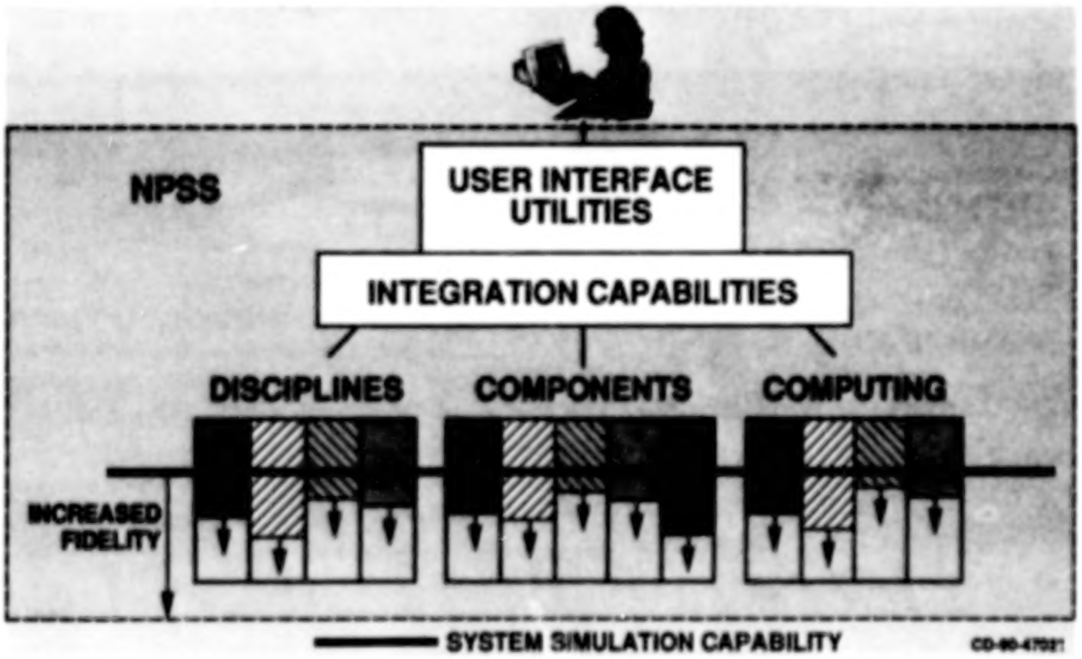
High-performance computing:
- Parallel processing
- Expert systems
- Interactive 3-D graphics
- Networks
- Data-base management systems
- Automated video displays

CD-91-84135

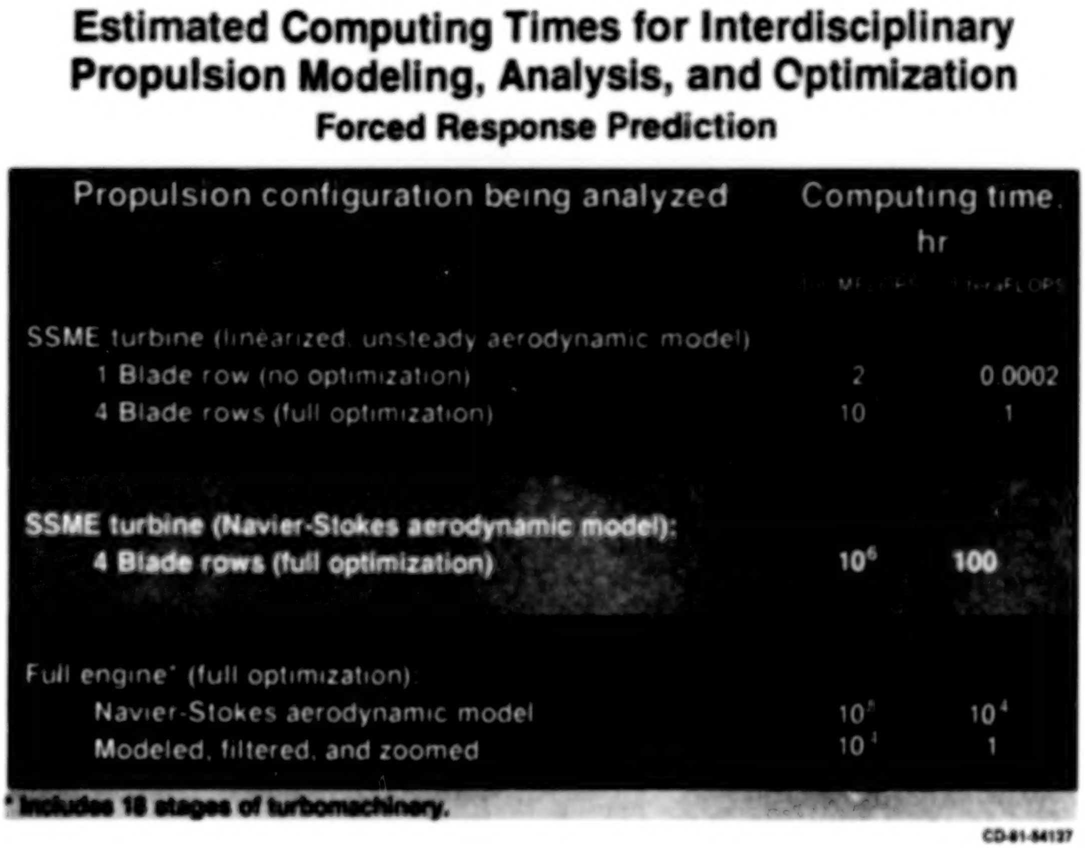
NPSS is a top-down systems approach that will provide designers with a tool to incorporate the relevant factors which affect system performance early in the design and analysis process when changes or modifications can be made relatively inexpensively. In terms of a propulsion system, such as an air-breathing gas turbine engine, this means coupling disciplines and components computationally to determine system attributes such as performance, reliability, stability, and life. Since these system attributes have traditionally been obtained in the test cell, NPSS is referred to as a "numerical test cell." A complete system analysis that includes multiple disciplines is a computationally intensive task. It requires a high-performance computing platform including both parallel and massively (i.e., more than 1000) parallel processors and a user interface that consists of expert systems, data-base management systems, and visualization tools.

The integrated, interdisciplinary system analysis requires advancements in the following technologies: (1) interdisciplinary analysis to couple the relevant disciplines such as aerodynamics, structures, heat transfer, chemistry, materials, and controls; (2) integrated system analysis to couple subsystems, components, and subcomponents at an appropriate level of detail; (3) a high-performance computing platform composed of a variety of architectures, including massively parallel processors, to provide the required computing speed and memory; and (4) a simulation environment that provides a user-friendly interface between the analyst and the multitude of complex codes and computing systems that will be required to perform the simulations.

NUMERICAL PROPULSION SYSTEM SIMULATION INTEGRATION


NPSS
USER INTERFACE UTILITIES
INTEGRATION CAPABILITIES
DISCIPLINES
COMPONENTS
COMPUTING
INCREASED FIDELITY
SYSTEM SIMULATION CAPABILITY
CD-90-47021

The implementation and integration of these technologies is a major challenge. The simulation environment and integration capabilities are depicted here. The system simulation is represented by the horizontal bar to signify that NPSS integrates into a system simulation the advancements that will continue to take place in the single-discipline, component, and computing fields. In this way NPSS will provide a focus for research and development in these fields. An additional challenge in NPSS will be the formation of interdisciplinary teams across NASA, industry, universities, and other Government agencies to develop and implement the needed technologies.



Estimated Computing Times for Interdisciplinary Propulsion Modeling, Analysis, and Optimization

Forced Response Prediction

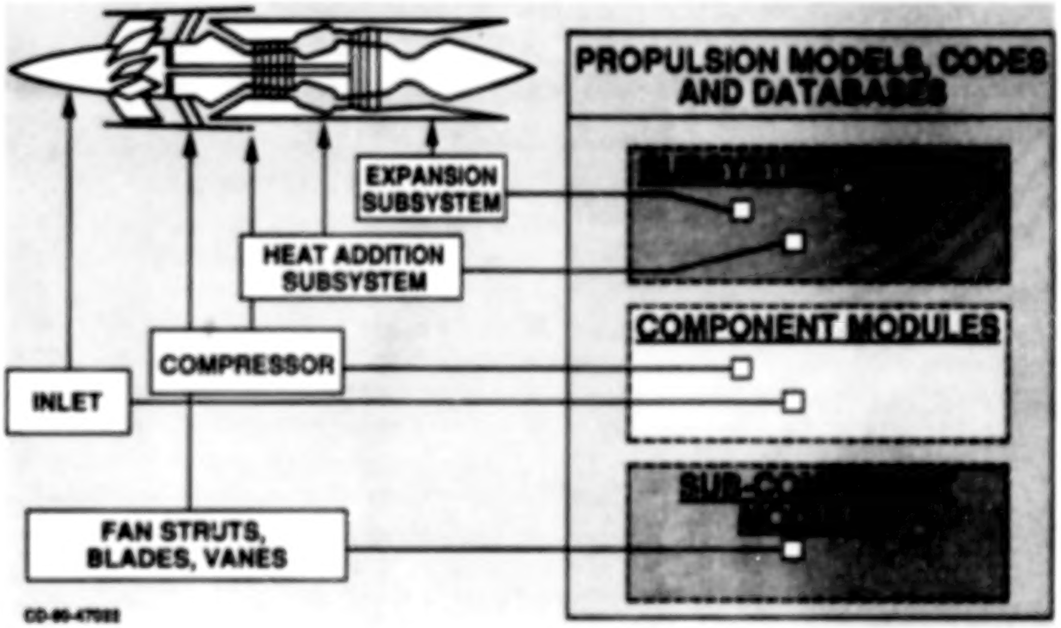
Propulsion configuration being analyzed	Computing time, hr	
	100 MFLOPS	1 teraFLOPS
SSME turbine (linearized, unsteady aerodynamic model)		
1 Blade row (no optimization)	2	0.0002
4 Blade rows (full optimization)	10	1
SSME turbine (Navier-Stokes aerodynamic model):		
4 Blade rows (full optimization)	**10^6**	**100**
Full engine* (full optimization)		
Navier-Stokes aerodynamic model	10^8	10^4
Modeled, filtered, and zoomed	10^4	1

* Includes 18 stages of turbomachinery.

CD-91-54137

The computational system simulations will be based on the view that only phenomena that affect system attributes, such as the life, reliability, performance, and stability of a propulsion system, are of interest to the designer or analyst. In addition, detailed analyses of an entire propulsion system will be so complex that even computers of teraFLOPS (10^{12} floating-point operations per second) speed will not be sufficient to perform cost-effective computations.

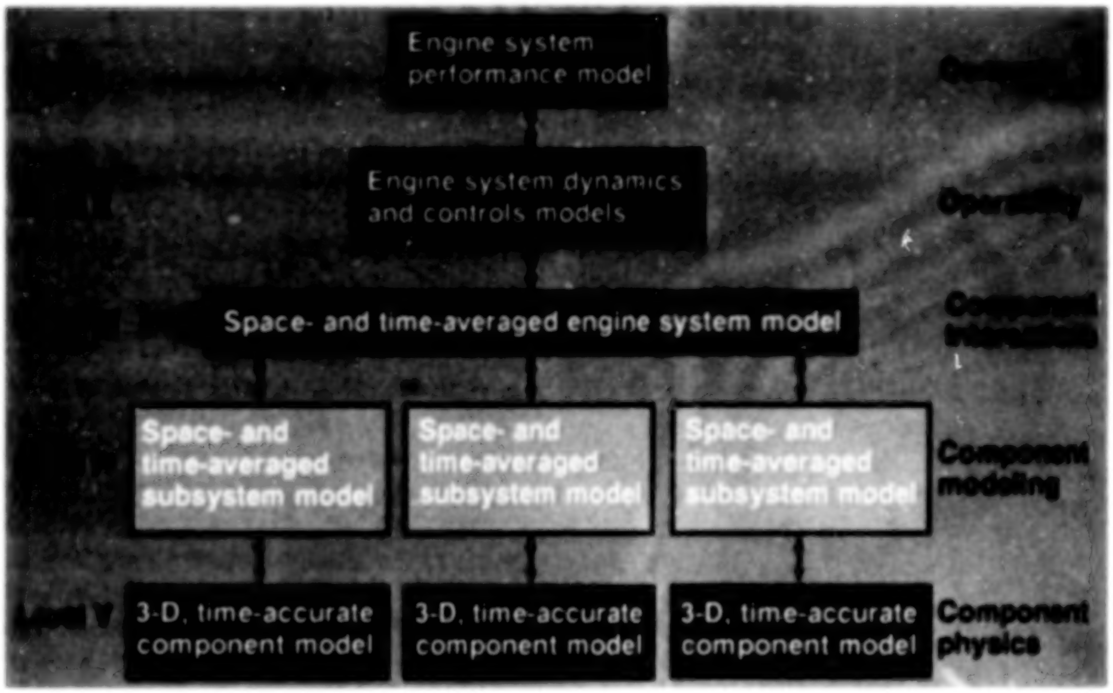
NUMERICAL PROPULSION SYSTEM SIMULATION
"ZOOMING IN" ON FAN EFFECTS


PROPULSION MODELS, CODES AND DATABASES
EXPANSION SUBSYSTEM
HEAT ADDITION SUBSYSTEM
COMPONENT MODULES
COMPRESSOR
INLET
SUB-CO
FAN STRUTS, BLADES, VANES

CD-89-47033

Consequently, a framework is being developed that will allow the physical processes resolved from a detailed analysis of a component or subcomponent to be communicated to a system analysis performed at a lower level of detail for purposes of evaluating system attributes. Conversely, the system analysis will provide the ability to evaluate which physical processes, occurring on the component and subcomponent level, are important to system performance. This will allow the engineer or scientist to focus or "zoom in" on the relevant processes within components or subcomponents. The zooming concept is depicted here. In this particular example a detailed analysis of the fan would be performed to study, for example, the effect of a new blade design on system performance. The inlet and compressor would be modeled at slightly lower levels of fidelity to resolve phenomena such an inlet distortion or upstream influences of the compressor blading. The combustor, the turbine, and the nozzle would be modeled in less detail, perhaps to determine shaft horsepower and thrust.

Hierarchy of Interdisciplinary Simulation Models



Engine system performance model

Engine system dynamics and controls models

Space- and time-averaged engine system model

Space- and time-averaged subsystem model

Space- and time-averaged subsystem model

Space- and time-averaged subsystem model

3-D, time-accurate component model

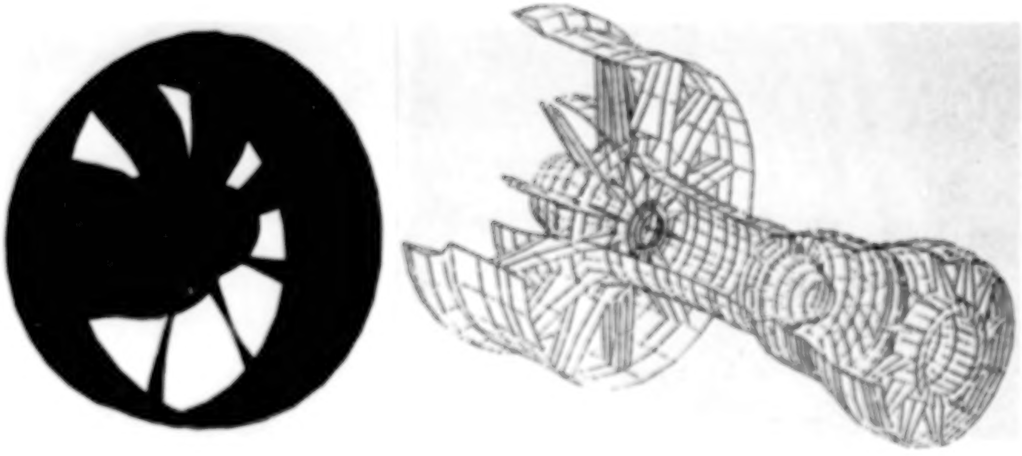
3-D, time-accurate component model

3-D, time-accurate component model

CD-91-84120

The implementation of the zooming approach requires a hierarchy of codes and models to be in place to provide a wide range of capabilities from detailed three-dimensional, transient analyses of components to time- and space-filtered analyses of the subsystems and systems. Modeling approaches will be developed for communicating information from a detailed analysis to a filtered analysis. Additional research will be required to understand the mechanisms by which phenomena on different length and time scales communicate. Research is already under way in computational fluid dynamics and structural mechanics to develop this modeling approach and will be extended to consider processes and scales appropriate for the entire propulsion system.

Single-Discipline Modeling



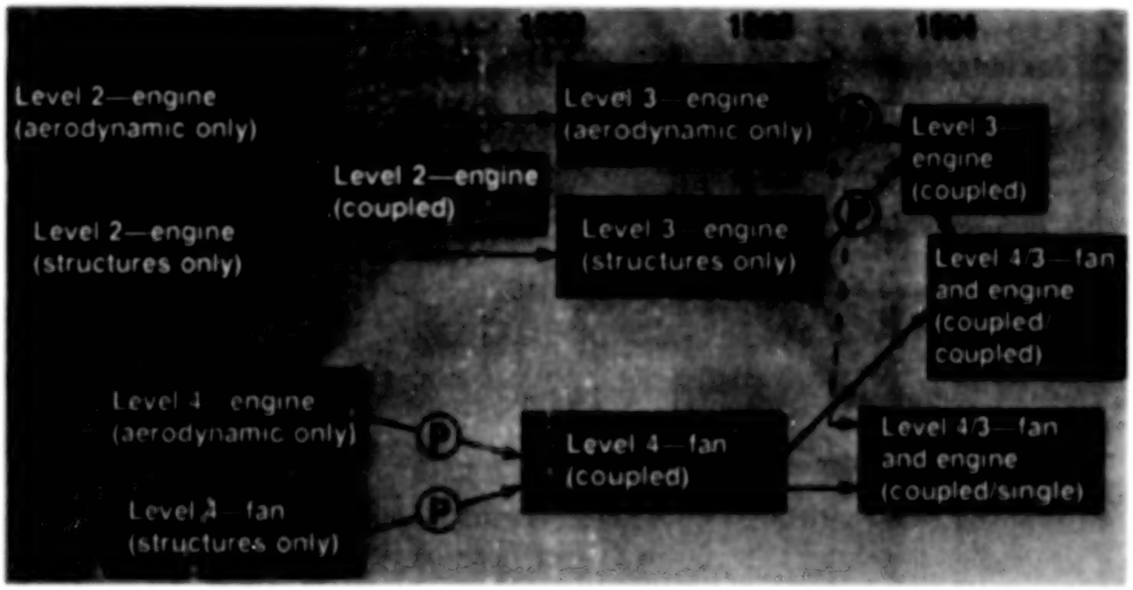
CD-91-54140

The fluid dynamic simulation model that will serve as the basis for the integrated system model will be the Adamczyk (ref. 1) average-passage formulation, which has been developed for multistage turbomachinery analysis. The average-passage model is based on the filtered forms of the Navier-Stokes and energy equations. This model was designed to resolve only the temporal and spatial scales that have a direct effect on the relevant physical processes. The effects of the unresolved scales, which appear as body forces and energy sources in the equations, are determined through semiempirical relations that are based on results from physical experiments or high-resolution numerical simulations. The results from the lower resolution analysis appear as boundary conditions for the high-resolution simulations. This model is currently applicable to time- and space-averaging of phenomena on the scale of the blade passing frequency and the passage size. Further development is required to extend the model for filtering in the presence of multiple scales and for other system components.

The structures modeling will be aimed at developing a comparable computational capability that will provide a means to traverse multiple scales of spatial resolution with a minimum number of variables at each level. In this way an analysis can proceed from a blade to a rotor to an engine core to the complete engine. The resulting system will have a minimum number of degrees of freedom consistent within the objectives of the analysis and will minimize the computational requirements. The methodology will be applicable to the solution of linear and incremental nonlinear analysis problems. This capability will be achieved through the formulation and implementation of a progressive substructuring technique.

NPSS Simulation Development Logic and Timing

Ⓟ Distribution or parallel implementation

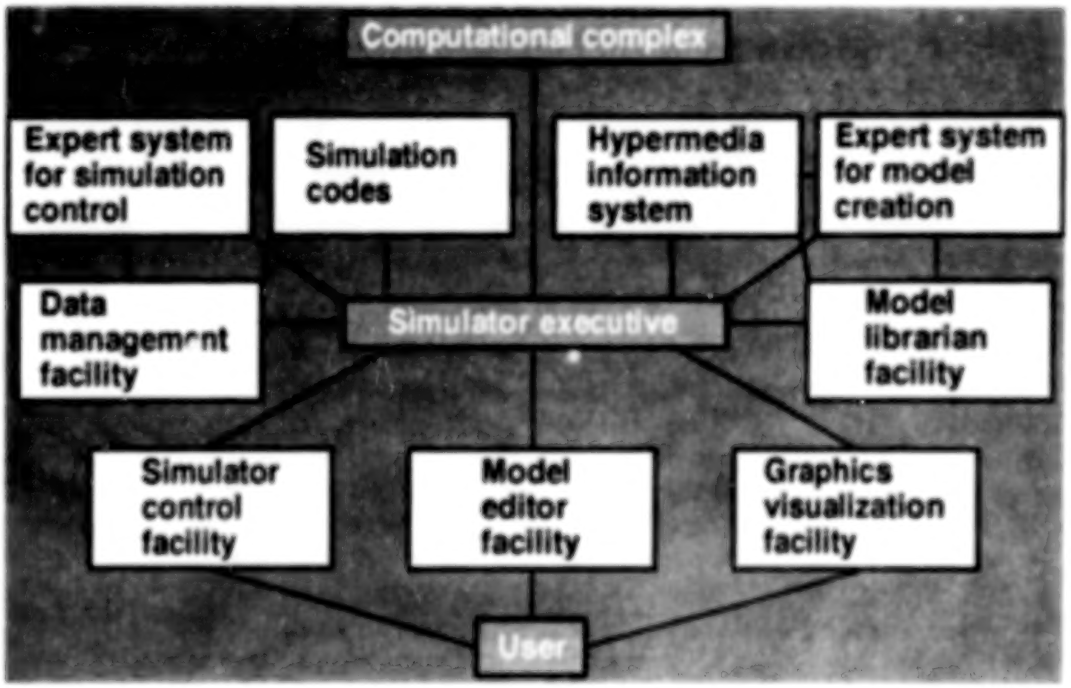

Level 2—engine (aerodynamic only)
Level 2—engine (structures only)
Level 2—engine (coupled)
Level 3—engine (aerodynamic only)
Level 3—engine (structures only)
Level 3—engine (coupled)
Level 4/3—fan and engine (coupled/coupled)
Level 4—engine (aerodynamic only)
Level 4—fan (structures only)
Ⓟ
Ⓟ
Level 4—fan (coupled)
Level 4/3—fan and engine (coupled/single)

CD-91-54141

The Lewis NPSS Program is now developing a level 2 (one-dimensional, time accurate) system code by coupling a fluid mechanics code, DIG-TEM (DIGital Turbine Engine Model), and a structures code, TETRA (Turbine Engine TRansient Analysis). The first application of this simulation capability will be on the Energy Efficient Engine (EEE), which was built and tested by the General Electric Company under contract to Lewis. We will "zoom in" on the fan and provide a level 4 (three-dimensional, time accurate) simulation of the fluid mechanics, structures, and ultimately acoustics phenomena. This level 4 simulation will initially be used to provide fan performance maps for use in the level 2 system simulation. The resulting simulation will be compared with the data obtained from the EEE test. Once verification of this technique is obtained, the simulation will be used to change the engine fan geometry (e.g., move the flow splitters to change the bypass ratio), to analyze the transient stability of the fan (blade deflections can change the flow characteristics in the fan), and to perform tradeoffs between engine efficiency and acoustic emissions.

The development of appropriate level 5 simulations will require new physical models, new multidisciplinary coupling methods, and new computational techniques. Future efforts will be directed toward developing a level 3 two-dimensional (i.e., axisymmetric), time-accurate model of propulsion systems that will make up the backbone of the NPSS software.

Simulator Architecture


Computational complex

Expert system for simulation control

Simulation codes

Hypermedia information system

Expert system for model creation

Data management facility

Simulator executive

Model librarian facility

Simulator control facility

Model editor facility

Graphics visualization facility

User

CD-91-54142

The capability for users to simulate, analyze, and optimize propulsion systems on high-performance, massively parallel computers will require extensive development of a user interface that will shield the user from the details of the computing system while providing sufficient guidance and assistance to perform the simulation at hand. The vision is that of a totally "seamless" environment. The environment consists of the integration of physical sciences, computer sciences, computer systems software, and computer systems hardware under the control of a global simulation executive. The computational simulation of multidiscipline, multicomponent problems entails a large number of variables that must be computed at multiple scales over multiple regions with results stored on local or global data-base environments. These types of problems can only be effectively solved by using massively parallel processors and networks where distributed parallel programming concepts can be readily implemented. Logic and software will be developed to adaptively allocate solution strategies and processors for a single discipline and for interdisciplinary analysis at both the local and global levels. Construction of simulations can be aided by a visual simulation editor coupled to an expert system "trained" in the use of the simulation codes. Artificial intelligence approaches, including expert systems and neural nets, will be investigated for assisting the user in making appropriate decisions in constructing a simulation. Advanced computer graphics, visualization, and animation complete this environment.

NPSS Architecture

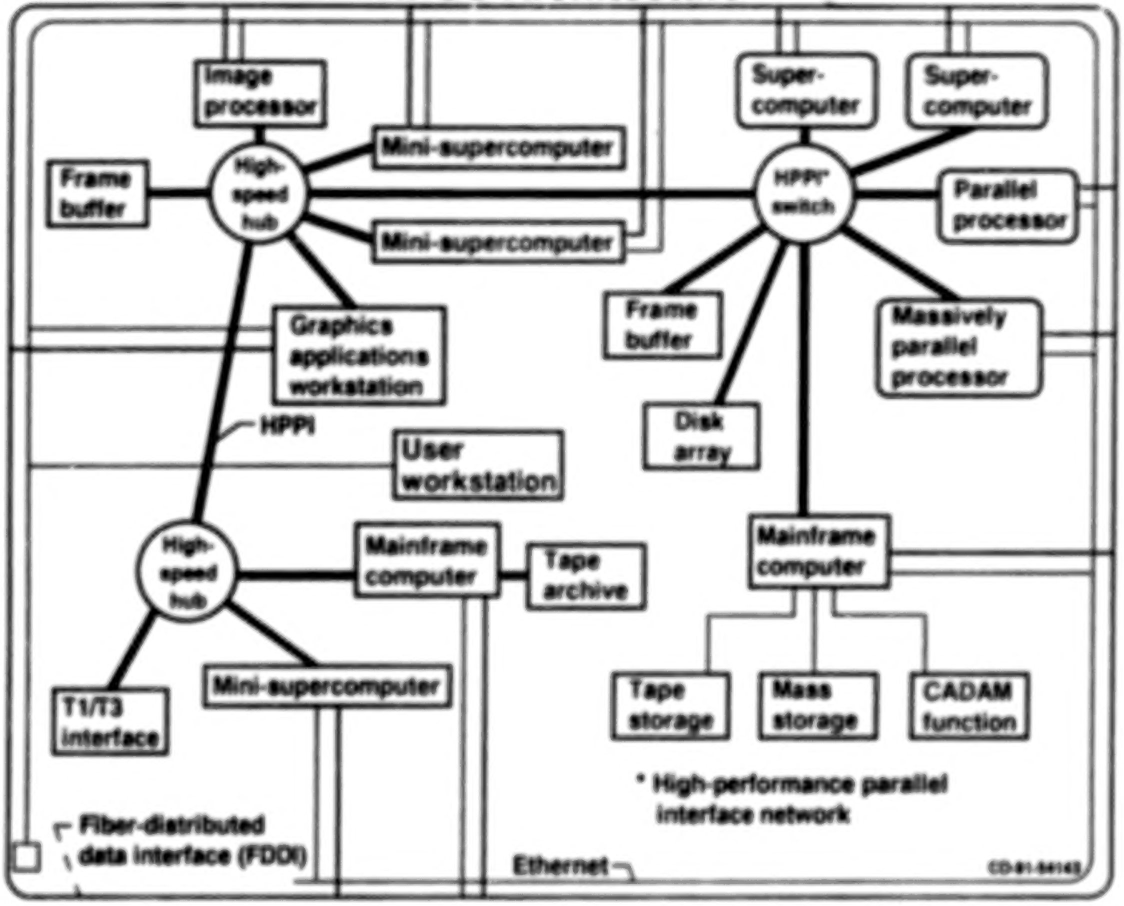


Image processor
Mini-supercomputer
Frame buffer
High-speed hub
Mini-supercomputer
Graphics applications workstation
HPPI
User workstation
High-speed hub
Mainframe computer
Tape archive
T1/T3 interface
Mini-supercomputer
Fiber-distributed data interface (FDDI)
Ethernet
Super-computer
Super-computer
HPPI* switch
Parallel processor
Frame buffer
Massively parallel processor
Disk array
Mainframe computer
Tape storage
Mass storage
CADAM function
* High-performance parallel interface network
CD-91-54143

It is expected that advances in parallel computing will make the integrated, interdisciplinary analysis of complex systems practical in design and analysis environments. At the same time it is expected that approaches to problem formulation and algorithm design will have to change to be able to exploit the new parallel architectures. Therefore, NPSS will establish a test-bed environment so that application and computer scientists can work closely together using state-of-the-art hardware and software tools to develop algorithms and to identify the appropriate computing architectures for propulsion system applications.

Computational Challenges in Propulsion Simulation

Nature of application	Critical computational technologies			
	Algorithms	User interfaces	Architecture	System software
Multiple disciplines	Tightly coupled; nonlinear	Visualization of multidisciplinary processes	Distributed heterogeneous	Large data bases; expert systems
Disparate scales	Hierarchical models; averaging/filtering; zooming	Visualization of multiscale processes	Reconfigurable; adaptive	Load-balancing; expert systems
Complex boundaries	Many-bodied real geometries	Interactive geometric modeling; grid generation	Reconfigurable; adaptive	Large data bases; load-balancing; expert systems
Engineering requirements	Robust, efficient, accurate, validated hierarchical models	Multipurpose tools; interfaces for simulation/ analysis/CADAM	Distributed heterogeneous; teraFLOPS and beyond	Interactive portable tools; expert systems; compatibility

CD-91-54184

The long-range goal of NPSS is to implement the shared-memory model, in either hardware or software, on massively parallel platforms. In the shared-memory model the programmer sees a uniform programming platform even though the hardware platform may consist of a variety of architectures such as cubes and rings. This not only simplifies the requirements for developing new code, but also provides the easiest, most flexible platform for the conversion of serial FORTRAN code, which proliferates throughout the computational engineering community today. The shared-memory model will allow this code to be used, without major revisions, for NPSS applications. Since the shared-memory model is a long-range objective, other technologies will be needed in the near term, including software tools for partitioning existing codes for solution on parallel computers.

Approach to NPSS

- Interdisciplinary analysis
- Propulsion system simulation at varying levels of detail (zooming)
- Uniform programming software standards
- Flexible, modular software architecture
- Propulsion system simulation on multiple available computing platforms
- Methodologies extended to massively parallel computational capability

CD-91-[illegible]

Products of NPSS Program

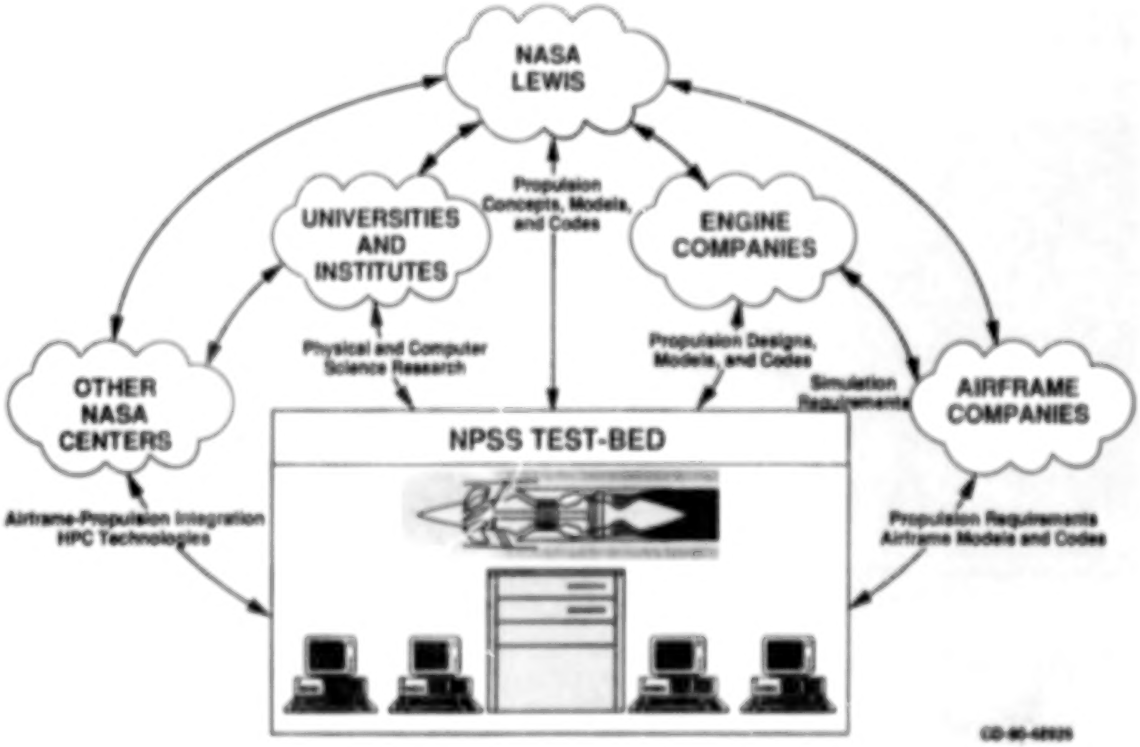
- Interdisciplinary models and algorithms for simulation and analysis of advanced propulsion system designs
- A simulation system architecture that integrates all of the required capabilities into a user-friendly computational environment
- High-performance computing technologies that enable rapid, interactive execution of the required algorithms and codes

[illegible]

SUMMARY

The Numerical Propulsion System Simulation is a long-range program with the ultimate goal of reducing the cost and time of developing advanced-technology propulsion systems. This goal will be achieved through a cooperative effort of NASA, industry, universities, and other Government agencies to develop the necessary technologies for integrating disciplines, components, and high-performance computing into a user-friendly simulation environment. The technologies associated with the physical sciences must include model development that reflects an understanding of the relevant physical processes rather than brute-force computational analysis. The computational algorithms must be developed in concert with the computing architectures to ensure efficient performance, particularly with parallel and massively parallel processors. In addition, a strong and effective management team will be needed to form the interdisciplinary teams from all organizations that will be required to define, advocate, and implement these technologies. Therefore, the IHPTET Program, being an experimental and demonstration program, will be able to provide the NPSS Program with valuable turbine engine data for use in validating the codes and models.

THE NPSS PARTNERSHIP


NASA LEWIS
UNIVERSITIES AND INSTITUTES
Propulsion Concepts, Models, and Codes
ENGINE COMPANIES
Physical and Computer Science Research
Propulsion Designs, Models, and Codes
Simulation Requirements
OTHER NASA CENTERS
AIRFRAME COMPANIES
NPSS TEST-BED
Airframe-Propulsion Integration
HPC Technologies
Propulsion Requirements
Airframe Models and Codes

REFERENCE

1. Adamczyk, J.J.: Model Equation for Simulating Flows in Multistage Turbomachinery. ASME 85-GT-226, 1985.

N91-20120

NASA'S AIRCRAFT ICING TECHNOLOGY PROGRAM

John J. Reinmann
NASA Lewis Research Center
Cleveland, Ohio

NASA's Aircraft Icing Technology program is aimed at developing innovative technologies for safe and efficient flight into forecasted icing. The program addresses the needs of all aircraft classes and supports both commercial and military applications. The program is guided by three key strategic objectives: (1) numerically simulate an aircraft's response to an in-flight icing encounter, (2) provide improved capabilities and techniques for ground and flight icing testing, and (3) offer innovative approaches to ice protection. Our fundamental research focuses on topics that directly support stated industry needs, and we work closely with industry to assure a rapid and smooth transfer of technology. This presentation reviews progress toward the three stated strategic objectives.

Program Charter

- **Develop innovative technologies for safe and efficient flight in icing conditions.**

CD-91-54435

NASA'S Aircraft Icing technology program is aimed at developing innovative technologies for safe and efficient flight into forecasted icing. The program addresses the needs of all aircraft classes and supports both commercial and military applications. Our fundamental research focuses on topics that directly support stated industry needs, and we work closely with industry to assure a rapid and smooth transfer of technology.

Program Strategic Objectives

- Numerical simulation of an aircraft's response to an in-flight icing encounter
- Improved capabilities and techniques for ground and in-flight icing testing
- Innovative approaches to ice protection

CD 91 54436

The icing technology program is guided by three strategic objectives: first, to develop and validate a system of computer codes that will numerically simulate an aircraft's response to an in-flight icing encounter; second, to provide experimental facilities that accurately simulate the natural icing environment and to develop new experimental capabilities and techniques to fully utilize these facilities; and third, to support the development and evaluation of advanced ice protection concepts that offer alternatives to compressor bleed air anti-icing systems.

Payoffs

Computer and experimental icing simulations

- Reduced certification time and costs
- Fewer hours of high risk flight testing
- Effect of icing on new aircraft designs

Advanced low power deicing systems

- Alternatives to compressor bleed air

CD 91 54437

The aircraft industry has repeatedly emphasized the payoffs they will realize from the NASA icing program. Validated computer codes and accurate experimental icing simulations will substantially reduce developmental and certification testing. This translates into reduced time and costs.

A complete numerical simulation of an aircraft's response to an icing encounter appears possible and economically feasible in the future.

Flight testing in icing conditions carries a high risk that will be reduced as numerical simulations become an acceptable alternative to some flight testing.

Accurate numerical simulations will allow earlier assessment of the effect of ice protection requirements on new aircraft designs. This assessment is especially important for future military aircraft with low observability, where ice protection must be considered in the initial design stages.

Next generation aircraft will be powered by advanced turbofan engines with higher bypass ratios and smaller core flows. Since the first priority for compressor bleed is cabin pressurization and air-conditioning, there may be inadequate bleed for conventional hot air anti-icing. Advanced, low-power deicers now under development may offer viable alternatives to conventional bleed systems.

Computer Codes & Validation

- Potential & viscous flow codes
- Water droplet trajectory codes
- Ice accretion codes
- Thermal deicer codes
- Airplane performance code

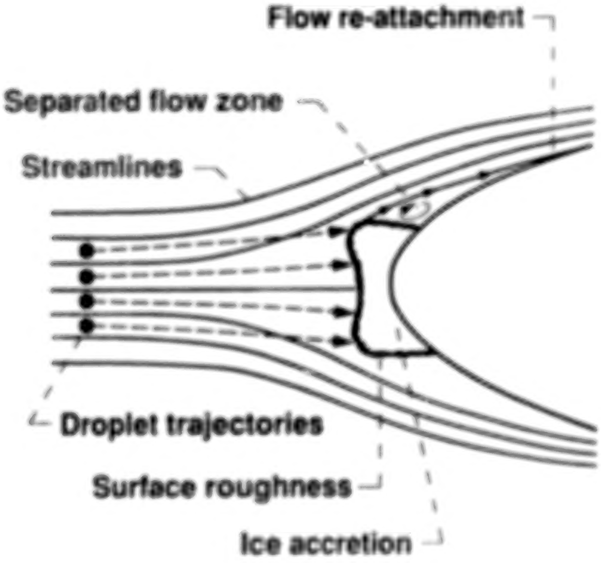
CD-91-[illegible]

This figure lists the basic codes in use or under development by NASA Lewis. These codes are being used, modified, or developed to conduct icing simulations on fixed-wing lifting surfaces, engine inlets, and rotor blades.

Industry uses these codes and their own modifications for preliminary design studies, aeroperformance predictions, design and analysis of proposed ice protection systems, and analytical support of the icing certification or qualification process.

Codes from this list will be integrated into a system of codes that will numerically simulate the complete response of aircraft to an in-flight icing encounter.

Ice Accretion Modeling/Validation


Flow re-attachment
Separated flow zone
Streamlines
Droplet trajectories
Surface roughness
Ice accretion

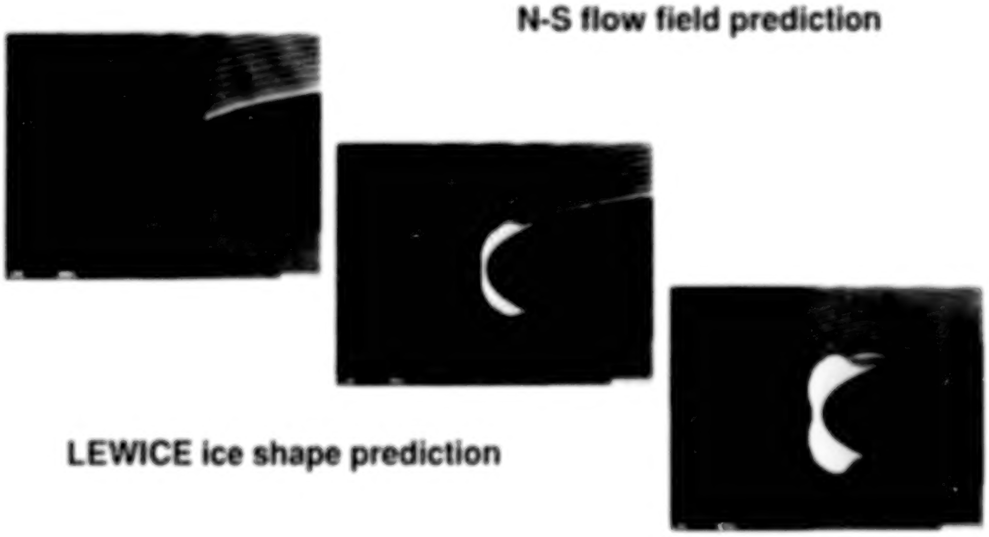
ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

The key physical processes that must be adequately modeled in any airfoil icing analysis methodology are illustrated on the left. These include proper modeling of the flow field around an airfoil with leading edge ice, which can cause flow separation and reattachment; water droplet impingement locations and flux; surface roughness; heat and mass transfer; and a thermodynamic energy balance for determining where ice forms.

Icing tunnels tests and flight tests in natural icing conditions provide actual ice shapes and drag data that can be used to validate code predictions. An airfoil under test in the NASA Icing Research Tunnel is shown on the right.

Ice Accretion/Aeroperformance Predictions

NACA 0012 airfoil, 0° AOA



CD-91-[illegible]

This figure shows predicted ice growths on an airfoil at sequential times during exposure to an icing cloud. These shapes were predicted with the NASA LEWICE two-dimensional ice accretion prediction code. LEWICE predictions agree well with ice formed on airfoils during icing tests in the Icing Research Tunnel (IRT) and also with ice formed on the Twin Otter icing research aircraft during flights in supercooled clouds.

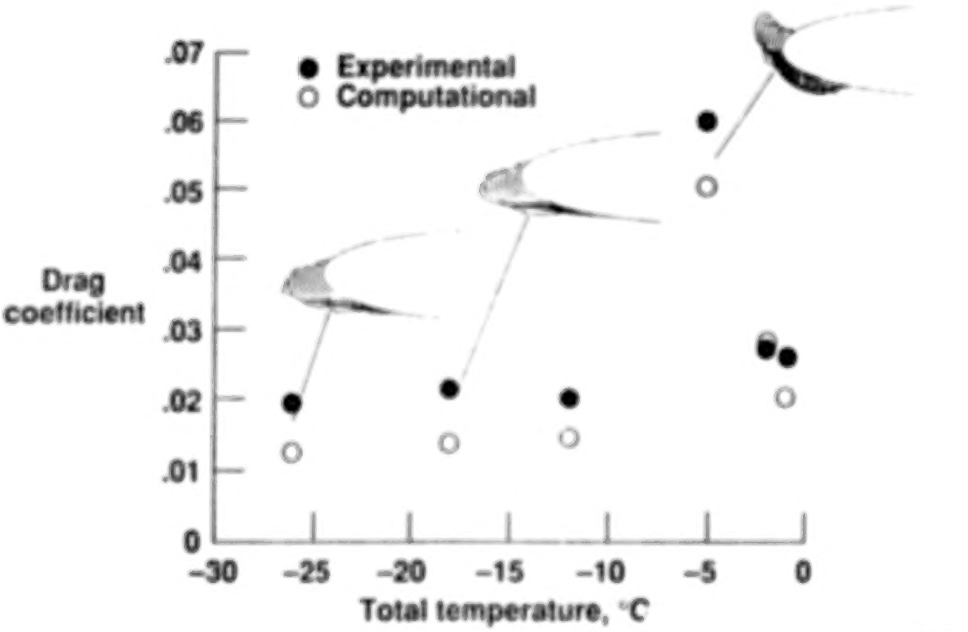
The LEWICE code has been distributed to over 35 organizations, and we continue to receive about one request per month.

The flow-field streamlines calculated with a Navier-Stokes solver are also shown here. Notice the separation bubble and reattachment zone behind the ice shape.

Ice Shapes and Drag Coefficients

Predicted Versus Experimental

Interactive boundary layer code upgrade of LEWICE

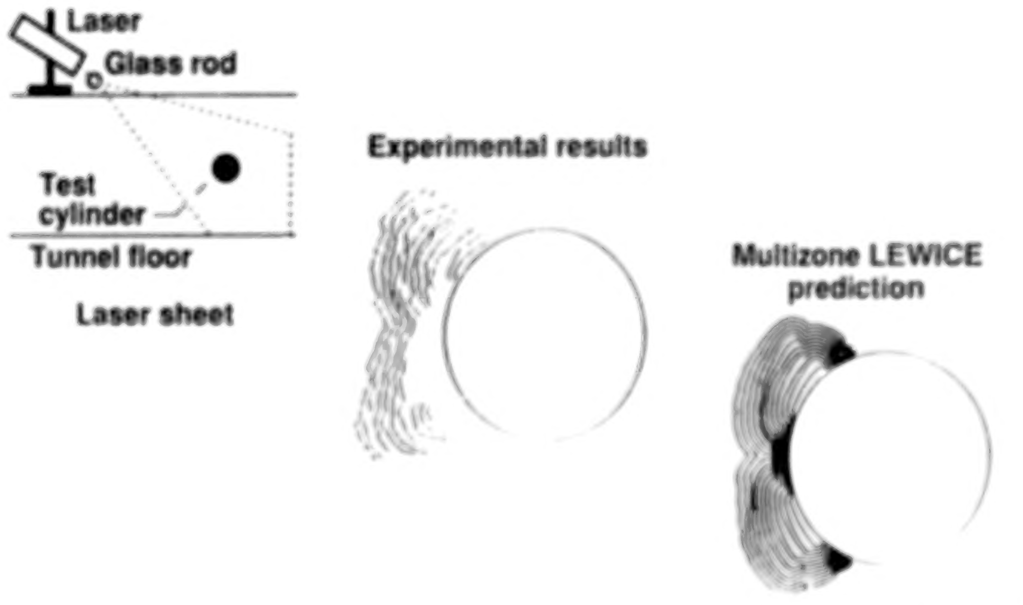


Ice on an airfoil causes decreased lift and $C_{L_{max}}$, decreased stall angle, increased stall speed, and increased drag. Thus, we need to predict not only the ice shape, but also its effect on aeroperformance. We have therefore upgraded the LEWICE code by incorporating an interactive boundary layer (IBL) code that calculates lift and drag changes.

This figure compares the LEWICE-IBL predictions for ice shape and drag with corresponding experimental results. Ice was accreted on a 21-in. chord, NACA 0012 airfoil in the IRT. Air temperature has a strong effect on ice shape and its resultant drag. Runs were made at several different air temperatures, while cloud conditions and airspeed were kept the same. The predicted and measured ice shapes agreed well, as did the drag coefficients. Especially encouraging was the ability to predict the dramatic drag increases observed experimentally at temperatures near -5 °C.

Multizone Roughness Model for LEWICE

Incorporated new routines in LEWICE that more closely model observed physics


Laser
Glass rod
Test cylinder
Tunnel floor
Laser sheet
Experimental results
Multizone LEWICE prediction

CD-91-[illegible]

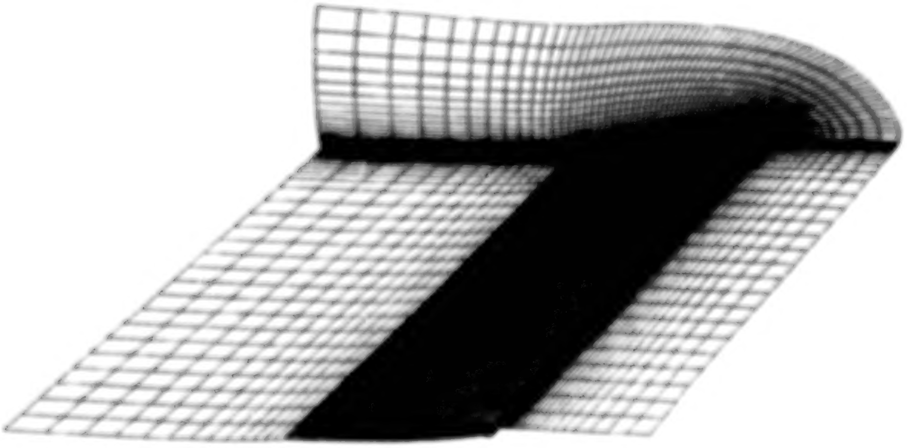
In our continuing efforts to upgrade and enhance the capabilities of LEWICE, we support fundamental studies on the physics of the ice accretion process.

This figure shows a test setup used to observe ice growth on a cylinder. By illuminating the surface of the ice with a laser sheet, we constructed a time history of the ice profile (shown in the middle diagram). This sequence of profiles suggested a multizone heat transfer model that was different from that used in LEWICE.

At the right is the ice shape predicted by a version of LEWICE modified to include the multizone heat transfer model. The analysis and experiment agree remarkably well.

This multizone model is undergoing further study and refinement, especially regarding surface roughness and its effect on heat transfer and transition location.

3D Iced Wing Aerodynamics



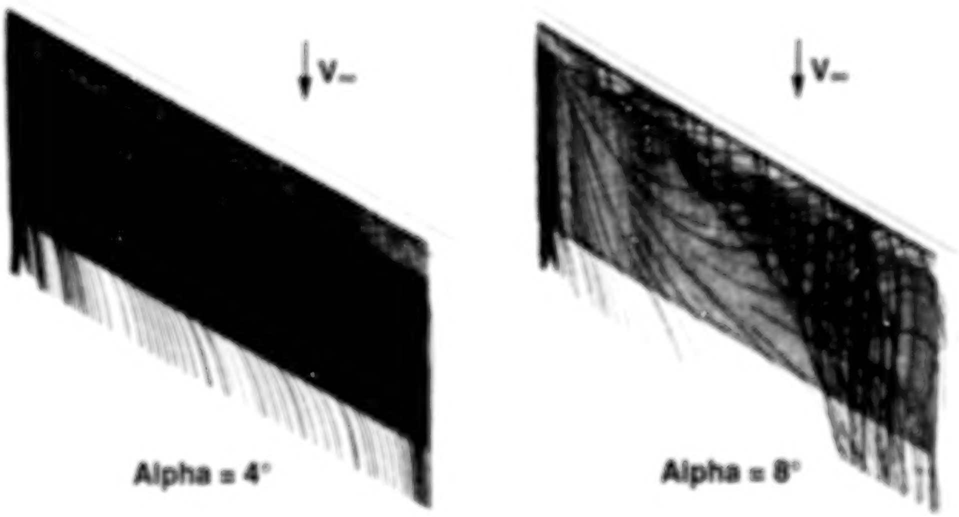
Grid for Navier-Stokes analysis of a swept, finite wing and vertical tunnel wall

CD-91-[illegible]

The aerodynamics of modern swept-wing aircraft is dominated by three-dimensional effects. NASA is therefore developing 3D flow codes that can model the flow over swept, finite-length wings with leading edge ice. This figure shows the geometry and gridding used in a Navier-Stokes analysis of a 30° sweep, finite-length wing attached to a vertical wall. The leading edge coordinates includes a leading edge ice shape.

This particular airfoil geometry has the same coordinate geometry as an actual model airfoil we are testing in a dry wind tunnel. The wind tunnel testing is designed to provide a comprehensive aerodynamic data base for validating 3D codes.

Navier–Stokes Analysis for Swept Wing with Leading Edge Ice



Particles trajectories for sweep angle of 30°

CD-91-[illegible]

One of the results from the Navier-Stokes analysis is shown in this figure as streamlines above the surface of the swept wing. The streamlines were divided into various color groups to help keep track of the flow.

At a 4° angle of attack, a small separation bubble exists behind the ice, but the flow reattaches. Near the leading edge, the flow has a strong spanwise component.

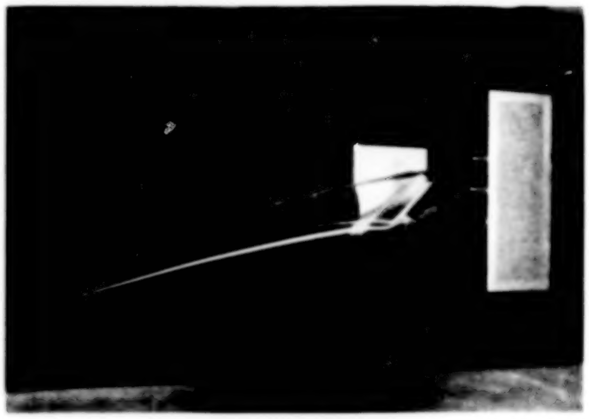
At an 8° angle of attack, the leading edge ice causes the flow to separate, and the resulting leading edge vortices have a strong spanwise component. Near the tip, the tip vortex interacts with the spanwise vortices to turn the flow toward the free-stream flow.

In the dry wind tunnel testing of this model, we will seed the flow with helium bubbles and use high-speed videography to observe the bubbles' trajectories. The experimental streamlines will be compared with the analytical predictions.

Dry Wind Tunnel Testing

Goal: Acquire an experimental data base for validating aeroperformance codes

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



Swept, finite wing with simulated ice on leading edge shown installed in wind tunnel

CD-91-54445

This figure shows a 30° swept wing model with simulated leading edge ice installed in a dry wind tunnel. The wind tunnel testing is designed to provide a comprehensive aerodynamic data base for validating 3D codes, such as the Navier-Stokes analysis and the 3D interactive boundary layer analysis.

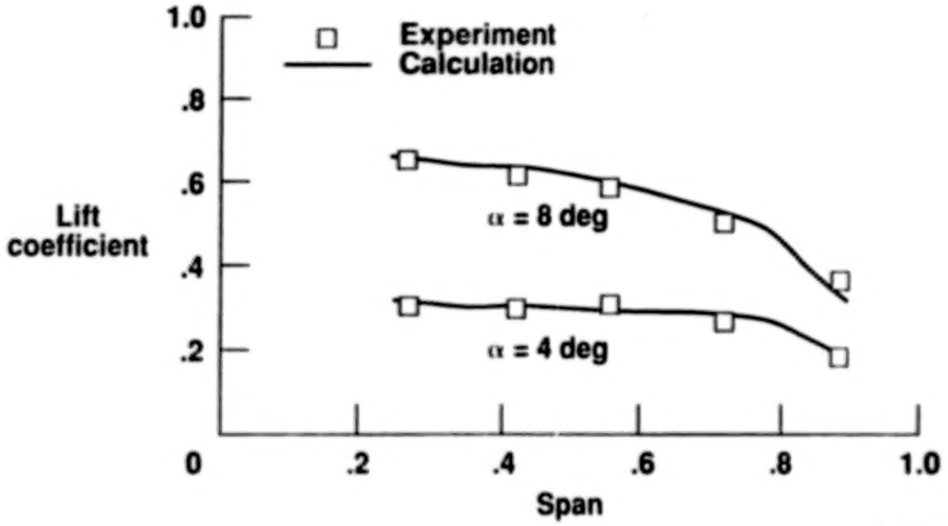
This model is heavily instrumented for surface pressures and is attached to a three-component force balance in the wind tunnel wall. Flow diagnostics include laser velocimetry and helium bubble seeding and tracking.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

C-1

Lift Coefficient Along Span of Iced Wing with 30° Leading Edge Sweep

Comparison between Navier-Stokes predictions and wind tunnel data



CD-91-54446

A comparison is shown here between the Navier-Stokes predictions and the wind tunnel results for lift coefficient versus span at 4° and 8° angles of attack. The agreement is nearly perfect.

Although the Navier-Stokes analysis is very accurate, it requires long run times on a supercomputer. An interactive boundary layer code coupled with a potential flow code is less accurate, but requires far less computational time. At this time, we are developing both approaches because the IBL approach is a good engineering design tool, and the Navier-Stokes approach is a good tool for modeling the detailed physics.

Airplane Performance in Icing

Goal

Develop a computer code that predicts performance and stability of modern aircraft with given ice contaminations on the lifting surfaces.

CD-91-54447

An important aspect of the numerical simulation of an aircraft's response to an icing encounter is the changes in aircraft performance and stability caused by ice. Thus, we are developing a computer code that predicts performance and stability of modern aircraft with given ice shapes on the lifting surfaces. This work will be carried out along with an ongoing effort to develop ice accretion codes for 3D surfaces. Later we will incorporate the ice accretion codes into the aircraft performance and stability code.

Airplane Performance in Icing

Surface pressure coefficients over NASA icing research aircraft
VSAERO code prediction

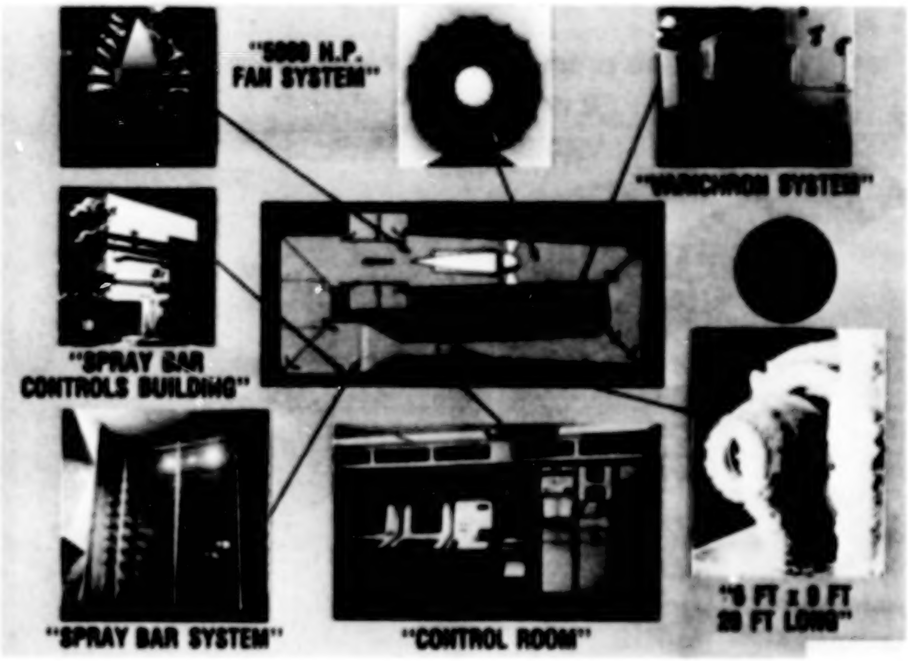
CD-91-54448

This figure shows the pressure distribution over the NASA Twin Otter icing research aircraft as calculated with an aircraft performance and stability code. Flight data from the Twin Otter will be used to validate the code at full-scale Reynolds numbers. First, the Twin Otter will be flown in dry air with "styrofoam" ice shapes on its tail surfaces, and later it will be flown in supercooled clouds.

In addition to the Twin Otter flight testing, we will conduct dry wind tunnel testing of a subscale model of a modern swept-wing aircraft with simulated ice on its lifting surfaces. The wind tunnel results will provide code validation data for a modern aircraft.

After we have acquired a good data base from the wind tunnel, we will use flight testing with a modern swept-wing aircraft to acquire a validation data base at full-scale Reynolds numbers.

Experimental Icing Simulation Techniques
NASA Icing Research Tunnel



CD-91-54449

In the area of icing facilities, NASA has two major commitments: first, to provide experimental facilities that accurately simulate the natural icing environment; and second, to develop new experimental capabilities and techniques that fully utilize these facilities.

The NASA Icing Research Tunnel and the NASA Twin Otter flight research aircraft are used extensively for code validation, advanced ice protection development, and actual component testing of commercial and military aircraft.

A schematic of the NASA Lewis Icing Research Tunnel (IRT) is shown in this figure. The components shown in the inserts are upgraded systems that were installed when the IRT was rehabilitated in 1986-87. The IRT is the largest refrigerated icing tunnel in the world. The test section is 6- x 9-ft, and 20 ft long. Two unique capabilities of the IRT are its spray nozzle system that creates the supercooled cloud and the heat exchanger and refrigeration plant that allow control of the air temperature down to -30 °F. In effect, we can "dial in the weather" any time of the year.

The IRT is one of NASA's busiest and most productive wind tunnels; in 1988 it logged 1330 hr of actual test time. It carries a 2-year backlog of work.

The IRT is used by NASA, the military, and industry for research and development testing. We conduct many joint programs with the military and with industry.

Experimental Icing Simulation Techniques Subscale Model Rotor Testing in the IRT

Scale model of UH-60 Blackhawk helicopter with 4 NACA 0012 blades, fully articulated rotorhead, and 6-component force balance

CD-91-54450

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Only one civilian helicopter, the French Super Puma, is certified in the United States for flight into forecasted icing conditions. It took 10 years of flight testing in natural icing to win that certification.

The U.S. rotorcraft industry estimates that, if flight testing in natural icing is the only acceptable means for certification, it would cost about 15 million dollars to certificate a helicopter. This cost is prohibitively high.

For several years, NASA and the U.S. rotorcraft industry have been engaged in a joint effort to develop new methods of reducing the cost and time needed to certify and qualify U.S. rotorcraft for icing.

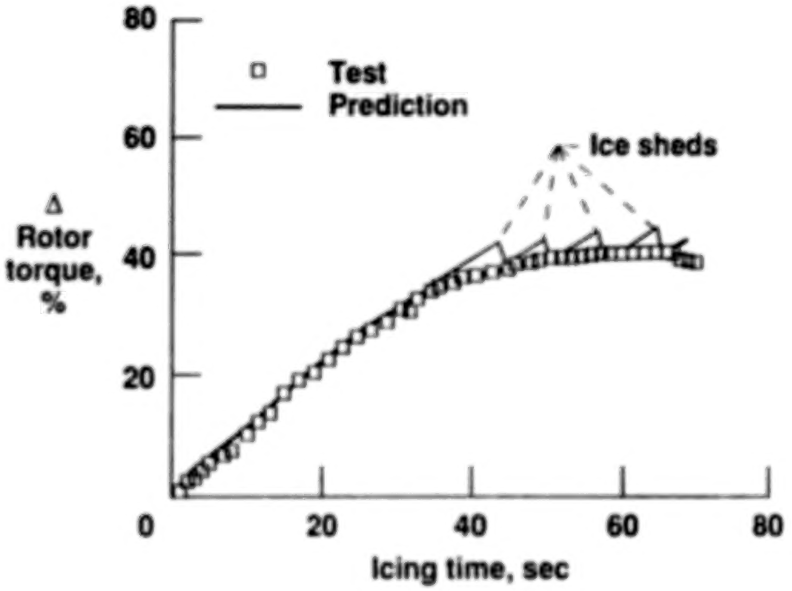
These methods include (1) computer codes that reliably predict full-scale rotor performance in icing and (2) experimental techniques for testing subscale model helicopter rotors in the IRT to acquire data for validating the codes and to develop a better understanding of the effects of icing on rotor performance.

The methods derived from this joint effort will also advance the state-of-the-art methods for predicting the effects of ice accretion and shedding for the Advanced Ducted Propeller and other thrusting devices.

This figure shows a subscale helicopter model being tested in the IRT. The model consists of a UH-60 Blackhawk helicopter fuselage, four NACA 0012 blades (5-in. chord, 6-ft diam), a fully articulated rotor head, and a six-component force balance housed under the Blackhawk fuselage.

Subscale Model Rotor Testing in the IRT

Rotor Torque vs Icing Time



CD-91-54451

This figure shows the torque rise caused by ice accretion on the rotors versus time in icing. The experimental results are compared with an analytical prediction developed by Sikorsky Aircraft. The rotor icing analysis includes an ice shedding model, which is necessary for good agreement with the experiment. The analysis also includes empirical airfoil performance-in-icing data that was acquired in another test program funded by NASA.

The comparison between theory and experiment, as shown here, is remarkably good for this particular test run. We also found similar agreement between theory and experiment for lift loss versus time in icing.

The four U.S. helicopter companies have received all the data from this test program. Selected results from this program, which include both analytical modeling and experimental validation, are being reported in several technical reports. The complete results will be published in a final contractor report.

We are also planning a second test entry into the IRT to expand the icing flight test envelope and to further study rotor performance at warmer outside air temperatures.

Advanced Low Power Ice Protection Systems

Goal

- Develop data base on low power deicers.

Approach

- Test eight different impulse deicer systems in NASA icing tunnel.
- Develop a structural analysis code for ice impact on engine fan blades.

CD 91 54052

NASA also supports the development and evaluation of advanced ice protection systems that offer lower-power alternatives to hot gas bleed and electrothermal heaters. The new class of electromechanical and pneumatic impulse deicers has stirred interest because they use only 10 to 20 percent of the power used by hot air systems.

Also of interest are some new, but largely untested, higher efficiency capillary tube thermal systems. We will not be discussing these systems, but NASA expects to test them in the future through collaborative efforts with industry.

Our current goal is to develop an experimental data base for the low-power impulse deicers. To that end, we have conducted a joint USAF/NASA/industry program to test eight different impulse deicer systems in the NASA IRT. We have also supported this effort through a Small Business Innovative Research Contract.

Since ice accumulates on the aircraft surfaces before the deicers are activated, ice particles will be shed from the surfaces during activation. If deicers are used on engine inlets, the engine must ingest ice particles without sustaining damage to fan blades or other components. For this reason, NASA has initiated the development of a structural analysis code for determining the response of engine fan blades to ice impact.

Air Force/NASA Low–Power Ice Protection Technology Program

NACA 0012 airfoil (21-in. chord) installed in IRT. Deicer systems were applied to the leading edge of the airfoil.

CD-91-54453

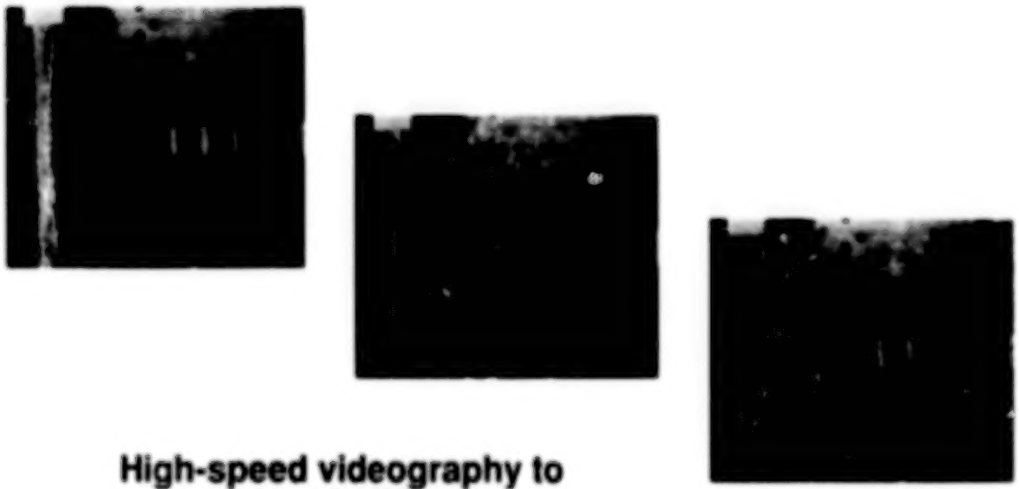
In the test program, eight different impulse deicer systems supplied by six companies were individually tested in the IRT under identical icing conditions. The systems employed pneumatic, eddy-current repulsion, or electro-expulsive means for ice removal.

This figure shows an airfoil with a deicer system installed on the leading edge. This airfoil geometry was chosen because its half-inch leading edge radius represented a challenge for most manufacturers, and because we are trying to simulate the small leading edge radii used on some military aircraft.

The deicers were characterized by four experimentally measured parameters: (1) maximum size of the shed ice particles; (2) amount, texture, and height of residual ice that remained on the surface after deicer activation; (3) energy per unit area required for one deicer activation; and (4) weight per unit area of deicer coverage. The deicers were tested under warm glaze ice conditions and cold rime conditions.

ORIGINAL PAGE IS
OF POOR QUALITY

Air Force/NASA Low–Power Ice Protection Technology Program



High-speed videography to capture ice shedding event

CD-91-54454

Our ability to capture an ice shedding event by means of high-speed videography is illustrated here. The video photos allowed us to estimate the size of the largest particles shed during an activation and to follow the ice breakup process in the airstream.

ORIGINAL PAGE IS
OF POOR QUALITY

Low–Power Deicer Technology

is Research
Center

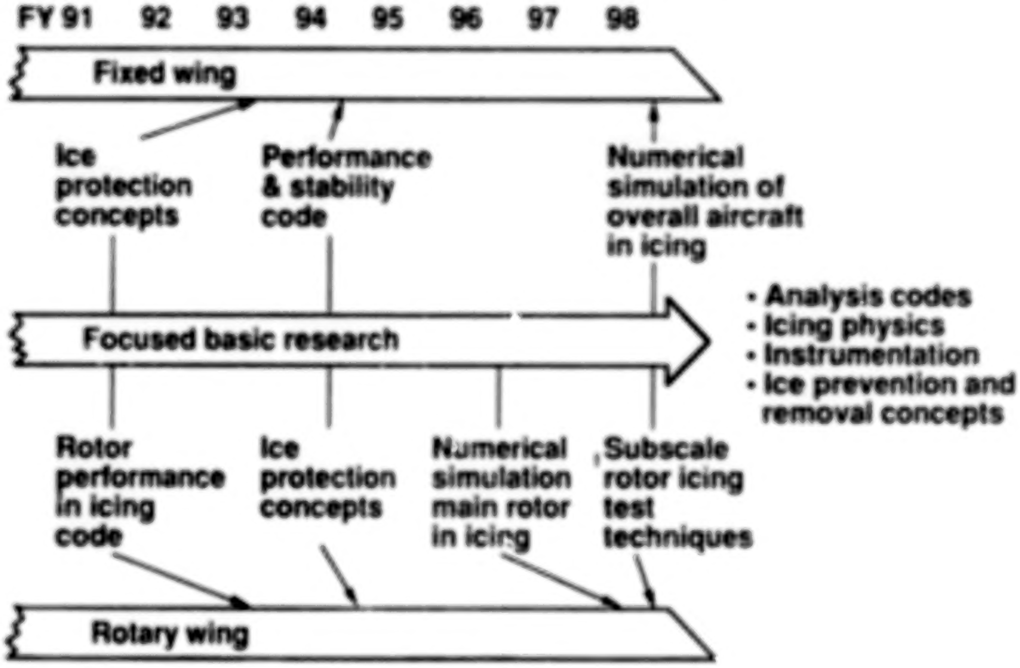
Eddy-current repulsion deicer undergoing development tests in IRT as part of a NASA SBIR contract

CD-91-54455

This figure shows ice being expelled from a cylinder by an eddy-current repulsion deicer strip that was undergoing development testing in the IRT. The work was done under a NASA Small Business Innovative Research Contract. This particular deicer is of interest to NASA because it can be applied over the outside of a component while causing only minimal disturbance to the inside of the component.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Aircraft Icing Technology Program


FY 91
92
93
94
95
96
97
98
Fixed wing
Ice protection concepts
Performance & stability code
Numerical simulation of overall aircraft in icing
Focused basic research
• Analysis codes
• Icing physics
• Instrumentation
• Ice prevention and removal concepts
Rotor performance in icing code
Ice protection concepts
Numerical simulation main rotor in icing
Subscale rotor icing test techniques
Rotary wing

The Aircraft Icing Technology Program has a strong, focused research effort supporting the strategic objectives for both fixed and rotary wing aircraft.

The various analytical codes that support ice accretion, aeroperformance, and ice protection are developed in the focused research. Companies and Government agencies receive these codes while they are still in the research stage so that NASA may get feedback on the user's experience with the codes and on desired additional capabilities.

Icing physics research supports the development of analytical models for ice accretion, shedding, and removal.

Droplet sizing instrumentation is essential for validating droplet trajectory codes and ice accretion codes.

While the strategic objectives are met in the outyears, codes, subscale model rotor-test techniques, and advanced ice protection concepts are continually worked throughout the program and promptly delivered to industry for inclusion in their own icing program.

CONCLUSION

The key strategic objectives of NASA's Aircraft Icing Technology Program are (1) to numerically simulate an aircraft's response to an in-flight icing encounter, (2) to provide improved capabilities and techniques for ground and flight icing testing, and (3) to offer innovative approaches to ice protection.

With a comprehensive computer code development program in place, we are progressing toward ultimately producing a methodology for numerically simulating the response of a complete aircraft to an icing encounter. At the same time, the codes are being used extensively by industry and Government in support of their icing programs.

Through a strong joint program with the U.S. helicopter industry, we have demonstrated that subscale model rotor testing in an icing wind tunnel provides valuable data for developing and validating computer codes that predict rotor performance in icing. The encouraging progress to date justifies further work in subscale model rotor testing in support of icing certification.

Through our joint USAF/NASA/industry test program we have succeeded in developing an extensive, but preliminary, data base on the new class of electromechanical and pneumatic impulse deicers. Because each impulse deicer needs a detailed evaluation under a wide range of icing conditions and under various operating modes, these systems will require much more testing.

Our good working relationships with industry, academia, and other Government agencies results in a combination of our individual resources, avoids duplication of effort and facilities, and expedites technology transfer to the user community.

N91-20121

RECENT ADVANCES IN LEWIS AEROPROPULSION FACILITIES

Frank J. Kutina, Jr.
NASA Lewis Research Center
Cleveland, Ohio

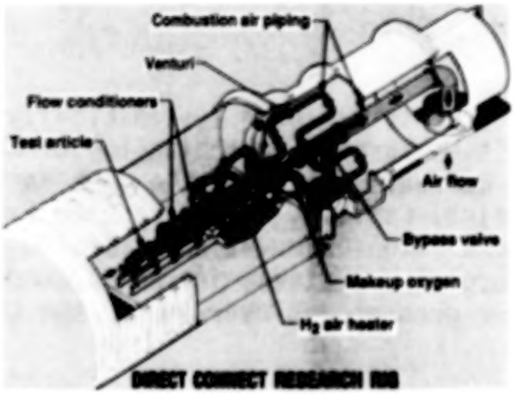
Guided by the aeropropulsion strategic plan, Lewis has been systematically refurbishing and upgrading its aeropropulsion facilities through a combination of R&D and construction of facilities (CoF) funding. Currently, much of the work is being accomplished as a part of the NASA Aeronautical Facility Revitalization Program. The total effort is addressing issues of improved flight simulation, improved productivity, capability to support advanced aeropropulsion systems, increased computational capability, and recertification. This paper presents an overview of the changes brought about by that effort.

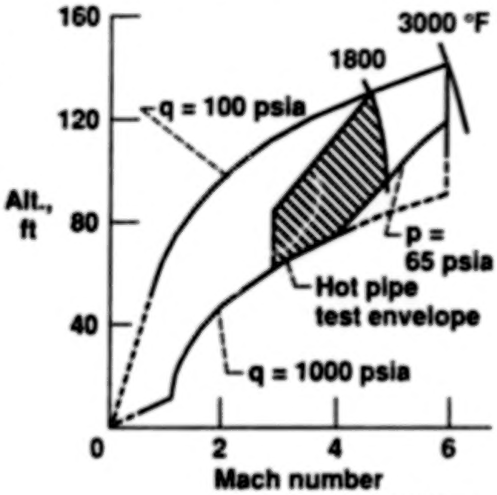
PSL-4 Hypersonic Modification

Operating Envelope

Hot pipe test conditions

- Temperature 70 – 1900 °F
- Pressure 10 – 65 psia
- Air flow 19 – 118 pps
- Hydrogen flow 0.05 – 0.68 pps


Combustion air piping
Venturi
Flow conditioners
Test article
Air flow
Bypass valve
Makeup oxygen
H_2 air heater
DIRECT CONNECT RESEARCH RIG



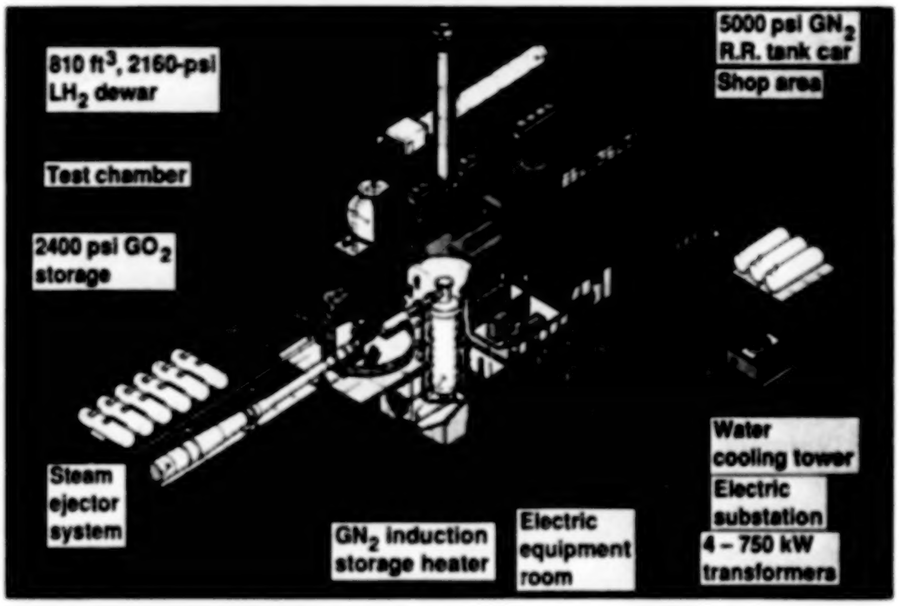
CD-91-54274

FLIGHT SIMULATION

A number of changes and additions have already been accomplished, and still more are in process, to provide a better simulation of higher Mach numbers. Figure 1 shows an addition made to the Propulsion Systems Laboratory cell 4, a full-scale turbine engine altitude cell. An in-line hydrogen fueled air heater was added to raise the model air supply (118 lb/sec) temperature to 3000 °F at 100 psia. This provides simulation up to Mach 6 in a direct-connect mode. Oxygen replenishment is provided to make up for depletion by the combustion process.

Plum Brook Hypersonic Tunnel Facility

Wind Tunnel Revitalization Program

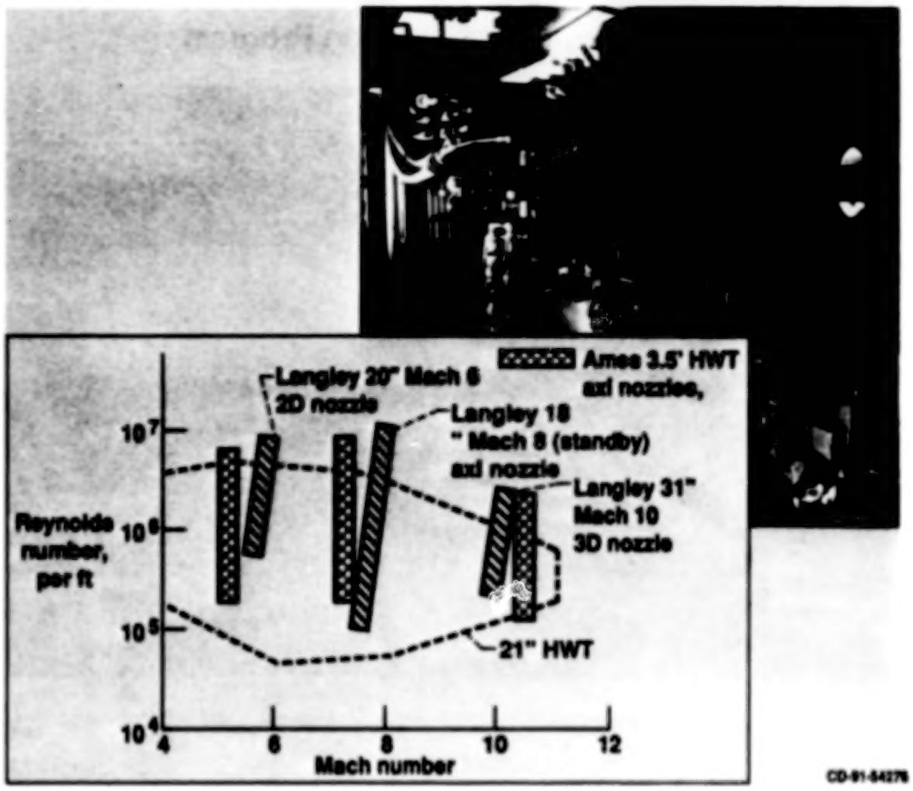


810 ft^3, 2160-psi LH$_2$ dewar
Test chamber
2400 psi GO$_2$ storage
Steam ejector system
GN$_2$ induction storage heater
Electric equipment room
5000 psi GN$_2$ R.R. tank car
Shop area
Water cooling tower
Electric substation
4 – 750 kW transformers

CD-91-54275

The Hypersonic Tunnel Facility (HTF) is being reactivated at Lewis' Plum Brook test site. This unique, indirectly heated, free-jet facility provides Mach 5, 6, and 7 test capability with a 42-in. nozzle. Because of the indirect heating, the air stream is not contaminated by combustion products and therefore provides a better simulation for propulsion testing. The test cabin can accommodate test articles up to 10 ft long, and a steam ejector system can provide altitude simulation to 125 000 ft. Final checkout tests for this facility are scheduled for early 1993.

21 - in. Hypersonic Wind Tunnel



CD-91-54276

In 1994, rebuilding and activation of a 21-in. hypersonic wind tunnel will begin, with IOC planned for 1996. This facility originally located at the Jet Propulsion Laboratory has a demonstrated superior flow quality and will be used initially to validate codes in the hypersonic regime.

Improvements are also being made in lower flight speed simulations. For example, in the 8- x 6-ft Supersonic Wind Tunnel/9- x 15-ft Low Speed Wind Tunnel complex, aerodynamic refinements are in process to improve the flow quality in the test sections. This work, to be completed in 1992, has the following goals:

Mach number	±0.001
Flow angularity	±0.1°
ΔV/V (8x6 test section)	0.25%

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Half-Span Model Installed in Icing Research Tunnel for Lift-Drag Tests

CD-91-54279

ORIGINAL PAGE IS
OF POOR QUALITY

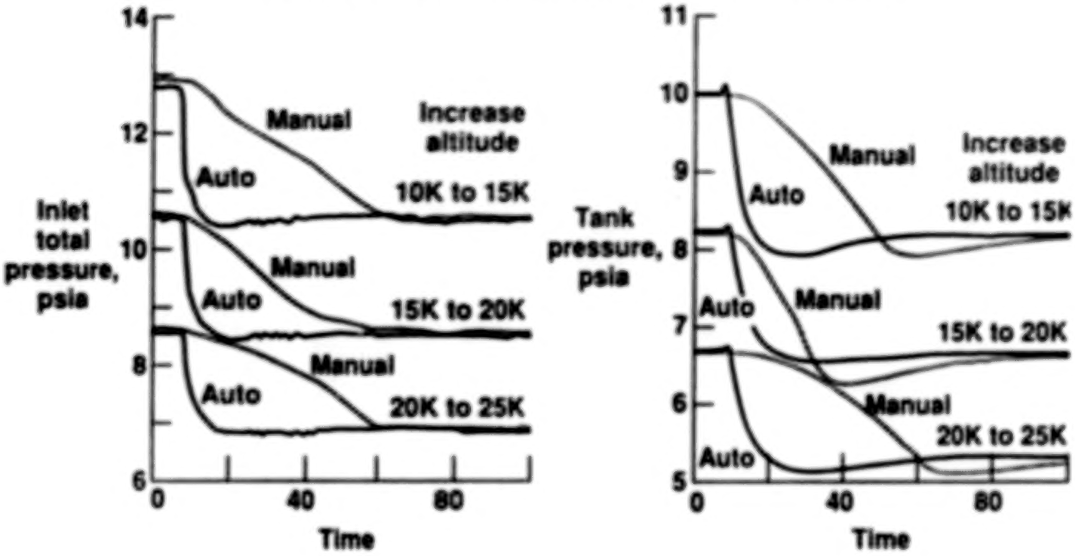
PRODUCTIVITY

The main thrust of NASA's Aeronautical Facility Revitalization Program is to refurbish NASA's aging aero facilities and improve their productivity and effectiveness. At Lewis, in addition to the general refurbishment, which should reduce unplanned outages, other planned changes and additions will improve productivity and test effectiveness. All major aero facilities will be equipped with separate model buildup and checkout rooms. This will allow models to be thoroughly debugged and prepped prior to installation in the test section, and will facilitate the handling of classified hardware. The control systems in these facilities are being upgraded to more modern, digital distributed control systems. These systems offer higher reliability, improved test point accuracy, reduced operator fatigue, and improved transient capability. An example of what these systems can do is shown above. The entire take-off roll, including rotation, of the full size aircraft was simulated in order to study the effects of deicing fluids on lift and drag. The tunnel air speed, temperature, and liquid water content and the angle of attack of the model were all preprogrammed in the control system, which provided very precise control and repeatable test conditions.

ORIGINAL PAGE

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Automated and Manual Control Responses in the Propulsion Systems Laboratory

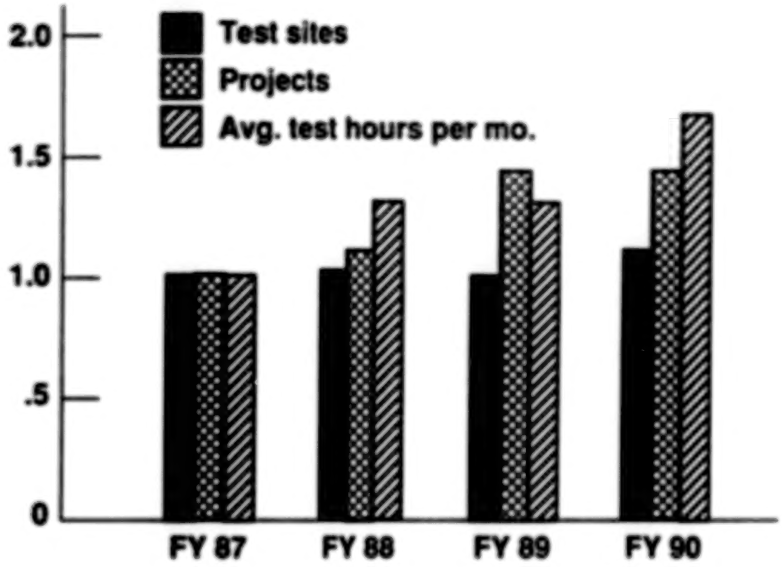


CD-91-54279

The figure depicts the improvements made in full scale turbine engine transient test capability. Shown are typical response times for the old, semiautomated system and what is possible with the new controls in terms of simulated flight Mach number and altitude.

The turbomachinery component cells have also benefited from the aeropropulsion facility general refurbishment. The old motor-generator variable frequency (VF) system, used to power variable speed drives for compressor testing, has been replaced with a new, solid-state VF system. The new VF system has greatly improved stability and resetability, providing an order of magnitude improvement in these parameters over the old system. The refrigerated air supply to the component test cells has been reactivated and updated to a turboexpander system capable of supplying air as low as -50 °F. This capability permits testing at high corrected speeds while at low mechanical speeds (and stresses).

Aeropropulsion Resource Utilization
(Normalized to FY '87)



CD-91-54291

The integrated effects on productivity for the major Lewis aeropropulsion facilities as a result of a number of incremental improvements are shown above. The data has been normalized to FY 1987 as a base year.

High Energy Fuel and Oxidizer Supply System in Aeropropulsion Facilities

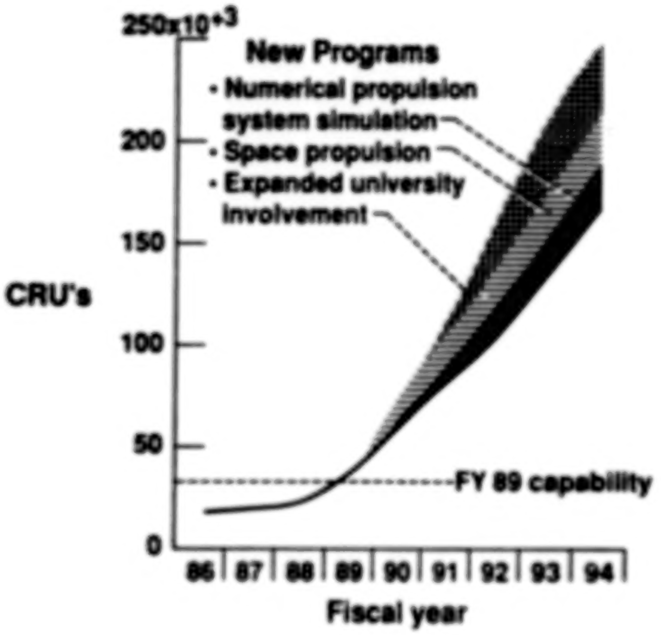
Facility	System and capability
10x10 Supersonic Wind Tunnel	GH_2 – 0.66 lb/sec at 1200 psia
8x6/9x15 Wind Tunnel	GH_2 – 1.0 lb/sec at 1500 psia
PSL	GH_2 – 2.75 lb/sec at 1000 psia GO_2 – 3.0 lb/sec at 100 psia 10.0 lb/sec at 300 psia
CE-9B	GH_2 – 0.12 lb/sec at 500 psia
PLF	GH_2 – 0.12 lb/sec at 1000 psia
HTF	GH_2 – 350,000 scf at 2400 psia GO_2 – 500,000 scf at 2400 psia LH_2 – 6000 gal

CD-91-54282

ADVANCED SYSTEMS

Lewis has long had a demonstrated capability to handle a variety of high energy propellant combinations safely in its rocket test cells. Drawing on this experience, hydrogen and oxygen supplies for the test article have been added to some of the aeropropulsion facilities to meet the needs of NASP and other high speed propulsion systems. A summary of these systems and their capabilities is shown above.

Supercomputing Requirements Based on Current and New Programs



CD-91-54283

COMPUTATIONAL CAPABILITY

Technology development at Lewis thoroughly integrates computation and experiment, with computing guiding experiment test planning and the experiment providing data for code validation. In 1988, Lewis undertook an assessment of its scientific computing needs. Several facts came to light:

(1) The ADP environment must be a balance between centralized and distributed computing systems in order for Lewis to do its job.

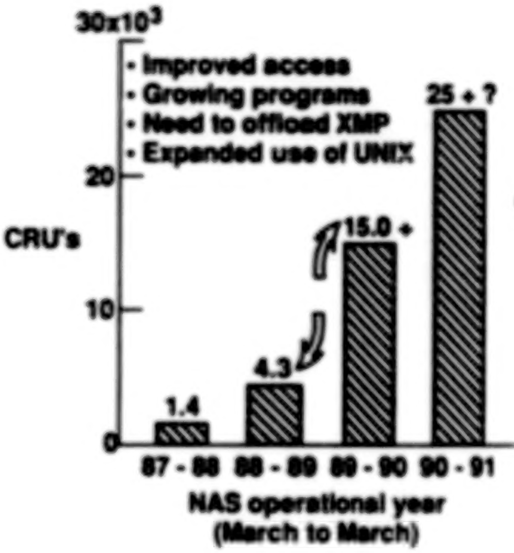
(2) There is a growing need for interactive graphics.

(3) Lewis was facing a saturation of its supercomputing capability.

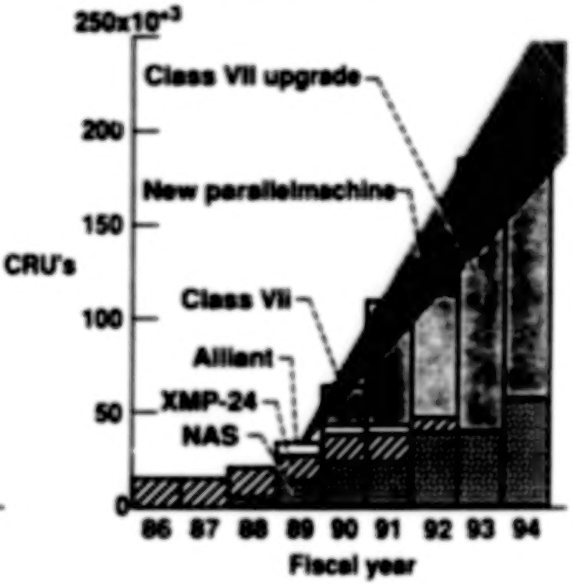
(4) There is a shortage of engineering workstations.

The saturation/shortfall in scientific computing is illustrated above.

Computational Facilities



Lewis Utilization of NAS

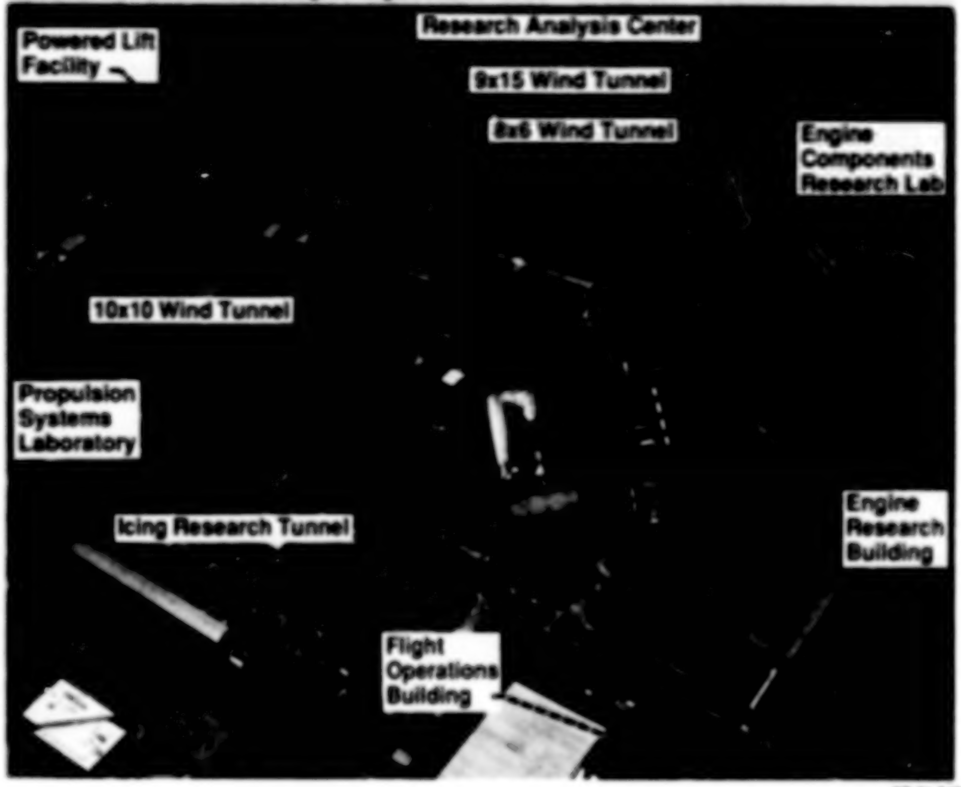


Meeting Lewis Supercomputing Requirements

CD-91-54284

In 1989, Lewis developed and implemented a long range plan to meet its computing needs. First, Lewis' share of the available time on the Numerical Aerodynamic Simulation (NAS) facility was increased. Second, a new class VII supercomputer was purchased and became operational in July 1990. This machine supplemented the existing CRAY XMP, and further upgrades are planned for the future. Third, buys of engineering workstations was increased. This plan, when fully implemented, should meet Lewis' scientific computing needs through the 1990's.

Aeropropulsion Facilities


Research Analysis Center
Powered Lift Facility
9x15 Wind Tunnel
8x6 Wind Tunnel
Engine Components Research Lab
10x10 Wind Tunnel
Propulsion Systems Laboratory
Icing Research Tunnel
Engine Research Building
Flight Operations Building

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

RECERTIFICATION

The average age of the Lewis aeropropulsion facilities is now very nearly 40 years. Many systems are unique and not built to ASME or other national consensus codes. In addition to the general refurbishment, requalification of the basic structures and supporting systems is necessary in order to assure continued availability. Lewis is now in its 10th year of a 15 year recertification program aimed at test facilities and their supporting systems. Facilities/systems have been prioritized as to criticality and potential hazard, and available funds are invested commensurate with that prioritization. The recertification of high pressure gas systems, pressure vessels, relief devices, flex hoses, and expansion joints is essentially complete. The recertification of the major aeropropulsion facilities is planned to coincide with other construction activities so as to minimize the programmatic impact.

CONCLUDING REMARKS

Lewis has positioned itself to have the required computational and experimental facilities to effectively undertake aeropropulsion research and technology development:

- Flow benches
- Laboratories
- Turbomachinery component rigs
- Full-scale turbine engine sea-level test stands
- Full-scale turbine engine altitude test stands
- Propulsion wind tunnels
- Aerodynamic wind tunnels
- Platform aircraft
- Desk-top, mid-size, and supercomputers

We stand ready to work cooperatively with industry, academia, and other Government agencies in our R&T efforts.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No.
NASA CP-10063

2. Government Accession No.

3. Recipient's Catalog No.

4. Title and Subtitle
Aeropropulsion '91

5. Report Date
March 1991

6. Performing Organization Code

7. Author(s)

8. Performing Organization Report No.
E-5954

10. Work Unit No.
505-62

9. Performing Organization Name and Address
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

11. Contract or Grant No.

13. Type of Report and Period Covered
Conference Publication

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546-0001

14. Sponsoring Agency Code

15. Supplementary Notes
Responsible person, David N. Bowditch, (216) 433-5729.

16. Abstract

In this 50th year since ground was broken for the NACA Aircraft Engine Research Laboratory, Aeropropulsion '91 provides a review of the recent accomplishments of the ongoing program at Lewis Research Center which has provided a foundation for the dominance of U.S. industry in aeropropulsion. The Lewis Aeropropulsion Program has the charter to develop advanced airbreathing propulsion technology for applications of national interest. To implement this program, Lewis concentrates its basic research in the critical disciplines of internal fluid mechanics, instrumentation and control, and structures and materials. These efforts are focused on developing new science, often implemented through advanced computer codes. Carefully designed experiments, often using new nonintrusive measurement techniques, provide the data required to validate the advanced codes. Recent results and planned accomplishments in these areas are reviewed. From this firm foundation, NASA Aeropropulsion Systems programs investigate technology for a number of applications including propulsion for subsonic and supersonic transports, high performance and hypersonic aircraft, rotorcraft, and general aviation. These programs are accomplished in cooperation with industry and are focused on validating advanced technology for the future engines of U.S. industry. These programs are reviewed with emphasis on accomplishments and future goals.

17. Key Words (Suggested by Author(s))
Aeropropulsion; Turbine engines; Aeronautical propulsion research; Materials; Structures; Internal fluid mechanics; Instrumentation; Controls; Subsonic propulsion; Supersonic propulsion; Hypersonic propulsion

18. Distribution Statement
Unclassified – Unlimited
Subject Category 07

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price*
Unclassified	Unclassified	575	A24

*For sale by the National Technical Information Service, Springfield, Virginia 22161

END

DATE

FILMED

MAY 23 1991